Frew

Version 19.2

Oasys

Oasys Ltd

13 Fitzroy Street
London
W1T 4BQ

Central Square
Forth Street
Newcastle Upon Tyne
NE1 3PL
Telephone: +44 (0) 191 238 7559
Facsimile: +44 (0) 191 238 7555

e-mail: oasys@arup.com
Website: http://www.oasys-software.com/

© Oasys Ltd. 2014

Frew Oasys GEO Suite for Windows

© Oasys Ltd. 2014

All rights reserved. No parts of this work may be reproduced in any form or by any means - graphic, electronic, or mechanical, including photocopying, recording, taping, or information storage and retrieval systems - without the written permission of the publisher.

Products that are referred to in this document may be either trademarks and/or registered trademarks of the respective owners. The publisher and the author make no claim to these trademarks.

While every precaution has been taken in the preparation of this document, the publisher and the author assume no responsibility for errors or omissions, or for damages resulting from the use of information contained in this document or from the use of programs and source code that may accompany it. In no event shall the publisher and the author be liable for any loss of profit or any other commercial damage caused or alleged to have been caused directly or indirectly by this document.

This document has been created to provide a guide for the use of the software. It does not provide engineering advice, nor is it a substitute for the use of standard references. The user is deemed to be conversant with standard engineering terms and codes of practice. It is the users responsibility to validate the program for the proposed design use and to select suitable input data.

Printed: May 2014

Table of Contents

1 About Frew 1

1.1 General Program Description ... 1

1.2 Program Features ... 1

1.3 Components of the User Interface ... 3

1.3.1 Working with the Gateway ... 3

2 Methods of Analysis 4

2.1 Stability Check ... 4

2.1.1 Fixed Earth Mechanisms ... 4

2.1.2 Free Earth Mechanisms ... 5

2.1.2.1 Multi-propped walls ... 5

2.1.3 Active and Passive Limits ... 8

2.1.4 Groundwater Flow ... 10

2.2 Full Analysis ... 11

2.3 Soil Models ... 12

2.3.1 Safe Method ... 13

2.3.2 Mindlin Method ... 13

2.3.3 Method of Sub-grade Reaction ... 14

2.4 Active and Passive Pressures ... 15

2.4.1 Effects of Excavation and Backfill ... 16

2.4.2 Calculation of Earth Pressure Coefficients ... 16

2.5 Total and Effective Stress ... 18

2.5.1 Drained Materials ... 19

2.5.2 Undrained Materials and Calculated Pore Pressures ... 19

2.5.3 Undrained Materials and User-defined Pore Pressure ... 21

2.5.4 Undrained to Drained Example ... 23

3 Input Data 23

3.1 Assembling Data ... 24

3.2 Preferences ... 29

3.3 New Model Wizard ... 30

3.3.1 New Model Wizard: Titles and Units ... 30

3.3.2 New Model Wizard: Basic Data ... 30

3.3.3 New Model Wizard: Stage Defaults ... 31

3.4 Global Data ... 32

3.4.1 Titles ... 32

3.4.1.1 Titles window - Bitmaps ... 33

3.4.2 Units ... 33

3.4.3 Material Properties ... 34

3.4.4 Nodes ... 37

3.4.5 Strut Properties ... 41

3.4.5.1 Modelling of Anchors ... 43

3.4.6 Surcharges ... 44

© Oasys Ltd. 2014

3.4.6.1 Application of Uniformly Distributed Loads ... 45
3.4.6.2 Application of Strip Loads ... 45
3.4.7 Partial Factors ... 47
3.4.8 Node generation data ... 50
3.5 Stage Data ... 51
3.5.1 Stage 0 - Initial Conditions ... 52
3.5.2 New Stages ... 52
3.5.3 Inserting Stages ... 53
3.5.4 Deleting a Stage ... 53
3.5.5 Editing Stage Data ... 53
3.5.6 Editing Stage Titles ... 54
3.5.7 Apply/Remove Surcharges ... 55
3.5.8 Insert/Remove Struts ... 55
3.5.9 Soil Zones ... 56
3.5.9.1 Dig/Fill Operations ... 59
3.5.10 Wall Data ... 60
3.5.11 Groundwater ... 62
3.5.12 Analysis Data ... 65
3.5.12.1 Model Type ... 66
3.5.12.2 Boundary Distances ... 67
3.5.12.3 Wall Relaxation ... 68
3.5.12.4 Fixed or Free Solution ... 68
3.5.12.5 Young's Modulus (E) ... 68
3.5.12.6 Redistribution of Pressures ... 69
3.5.12.7 Minimum Equivalent Fluid Pressure ... 69
3.5.12.8 Passive Softening ... 71
3.5.13 Convergence Control ... 72
3.5.13.1 Maximum number of Iterations ... 73
3.5.13.2 Tolerance for Displacement ... 73
3.5.13.3 Tolerance for Pressure ... 73
3.5.13.4 Damping Coefficient ... 73
3.5.13.5 Maximum Incremental Displacement ... 74

4 Frew-Safe Link 74
4.1 Data Entry ... 74
4.2 Data Conversion ... 79
4.2.1 Stages/Runs ... 79
4.2.2 Geometry ... 81
4.2.3 Restraints ... 81
4.2.4 Surcharges ... 82
4.2.5 Struts ... 82
4.2.6 Materials ... 83
4.2.6.1 First Stage Material ... 85
4.2.7 Groundwater ... 87
4.2.8 Unsupported Features ... 88

5 Integral Bridge Analysis 88
5.1 Data Entry ... 89
5.2 Algorithms ... 93

6 Seismic Analysis 93

© Oasys Ltd. 2014

6.1 Data Entry 94
6.2 Seismic Analysis Methods 96
6.2.1 Calculation of Seismic Soil Pressure Coefficient 96
6.2.2 Wood's Method 96
6.2.3 Mononobe-Okabe Method 97
6.2.4 Load Application Methods 100
6.2.5 Groundwater Loading 101

7 Output 101
7.1 Analysis and Data Checking 101
7.2 Tabulated Output 104
7.2.1 Stability Check Results 106
7.2.2 Detailed Results 107
7.2.2.1 Results Annotations and Error Messages 108
7.3 Graphical Output 109
7.4 Batch Plotting 112

8 Detailed Processes in Frew 113
8.1 General 113
8.2 Approximations Used in the Safe Method 113
8.2.1 The Basic Safe Model 113
8.2.2 Application of the Model in Frew 114
8.2.3 Accuracy with Respect to Young's Modulus (E) 116
8.2.3.1 Linear Profile of E With Non-Zero Value at the Surface 116
8.2.3.2 Irregular Variation of E 116
8.2.4 Effect of the Distance to Vertical Rigid Boundaries 118
8.2.4.1 Accuracy of Modelling Boundaries in Frew 119
8.2.5 Friction at the Soil/Wall Interface 122
8.2.5.1 Accuracy of the 'Fixed' Solution 123
8.3 Approximations Used in the Mindlin Method 123
8.3.1 The Basic Mindlin Model 123
8.3.2 Application of the Model in Frew 124
8.3.2.1 Accuracy of the Mindlin Solution in Frew 124
8.4 Calculation of Active and Passive Limits and Application of Redistribution 126
8.4.1 General 127
8.4.2 Application in Frew 128
8.4.3 Iterative Technique Adopted in Frew 132
8.5 Active Pressures Due to Strip Load Surcharges 133
8.5.1 Application in Frew 133
8.5.2 Passive Pressures Due to Strip Load Surcharges 135
8.5.2.1 Requirement 1 136
8.5.2.2 Requirement 2 137
8.5.2.3 Requirement 3 137
8.5.2.4 Requirement 4 138
8.6 Wall and Strut Stiffness Matrices 139
8.6.1 Wall Stiffness Matrices 139
8.6.2 Strut or Anchor Matrices 141
8.7 Modelling Axi-symmetric Problems Using Frew 142
8.7.1 Soil Inside the Excavation 143

© Oasys Ltd. 2014

8.7.2 Soil Outside the Excavation 144
8.7.3 Stiffness Varying with Depth 145
8.8 Modelling Berms 145
8.8.1 Rigorous Method 146
8.8.2 Simplified Procedure 148
8.9 Creep and Relaxation 149
8.9.1 Changing From Short Term to Long Term Stiffness 149
8.10 Undrained to drained behaviour - Manual Process 151
8.10.1 Undrained to Drained Examples 152
9 List of References 158
9.1 References 158
10 Brief Technical Description 159
10.1 Suggested Description for Use in Memos/Letters, etc 159
10.2 Brief Description for Inclusion in Reports 159
11 Manual Example 160
11.1 General 160
Index 161

© Oasys Ltd. 2014

1 About Frew

1.1 General Program Description

Frew (Flexible REtaining Walls) is a program that analyses flexible earth retaining structures such as sheet pile and diaphragm walls. The program enables the user to study the deformations of, and stresses within, the structure through a specified sequence of construction.

This sequence usually involves the initial installation of the wall followed by a series of activities such as variations of soil levels and water pressures, the insertion or removal of struts or ground anchors and the application of surcharges.

The program calculates displacements, earth pressures, bending moments, shear forces and strut (or anchor) forces occurring during each stage in construction.

It is important to realise that **Frew** is an advanced program analysing a complex problem and the user must be fully aware of the various methods of analysis, requirements and limitations discussed in this help file before use.

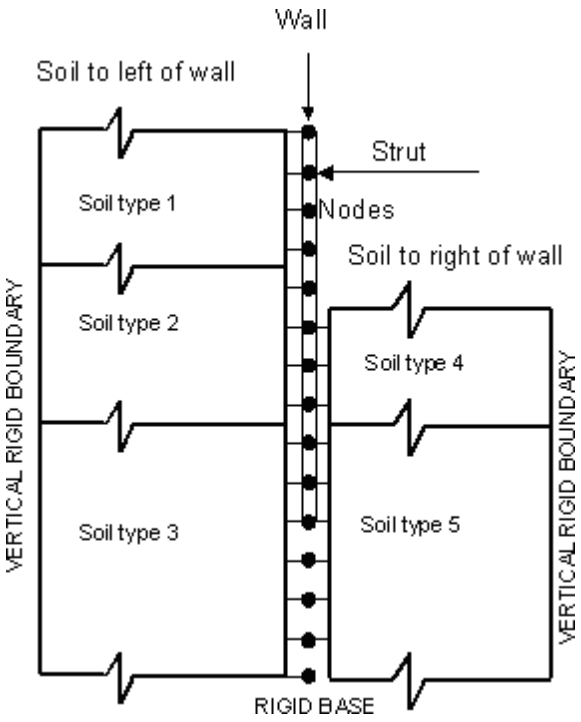
The program input is fully interactive and allows both experienced and inexperienced users to control the program operation.

1.2 Program Features

The main features of **Frew** are summarised below:

- The **geometry** of the wall is specified by a number of nodes. The positions of these nodes are expressed by reduced levels. The nodes can be generated from the other data (soil interface levels etc.) using the Automatic Node Generation feature.

© Oasys Ltd. 2014



- **Wall stiffness** is constant between nodes, but may change at nodes. The base of the wall may be specified at any node, nodes below this are in "free" soil. The wall stiffness can be changed or relaxed at the various stages of the analysis.
- **Soil** profiles are represented by a series of horizontal soil strata that may be different each side of the wall. The boundaries of soil strata are always located midway between node levels. This constraint will be accommodated when using the Automatic Node Generation feature.
- **Struts** may be inserted and subsequently removed. Each strut acts at a node. If the Automatic Node Generation feature is used, a node will be generated at each specified strut level. A strut may have a specified stiffness, pre-stress and lever arm and may be inclined to the horizontal. For inclined struts with a non-zero lever arm, a rotational stiffness at the node is modelled.
- **Surcharges** may be inserted and subsequently removed. Each surcharge comprises a uniformly distributed load or a pressure load of a specified width.
- Soil may be **excavated, backfilled** or changed at each stage, on either side of the wall.
- **Water** pressures may be either hydrostatic or piezometric.
- The program provides a selection of **stiffness models** to represent the soil.
    1. "**Safe**" flexibility model.
    2. Mindlin model.
    3. Sub-grade reaction model.

Note: The **sub-grade reaction model** is currently not active, it will be added to **Frew** in the near future.

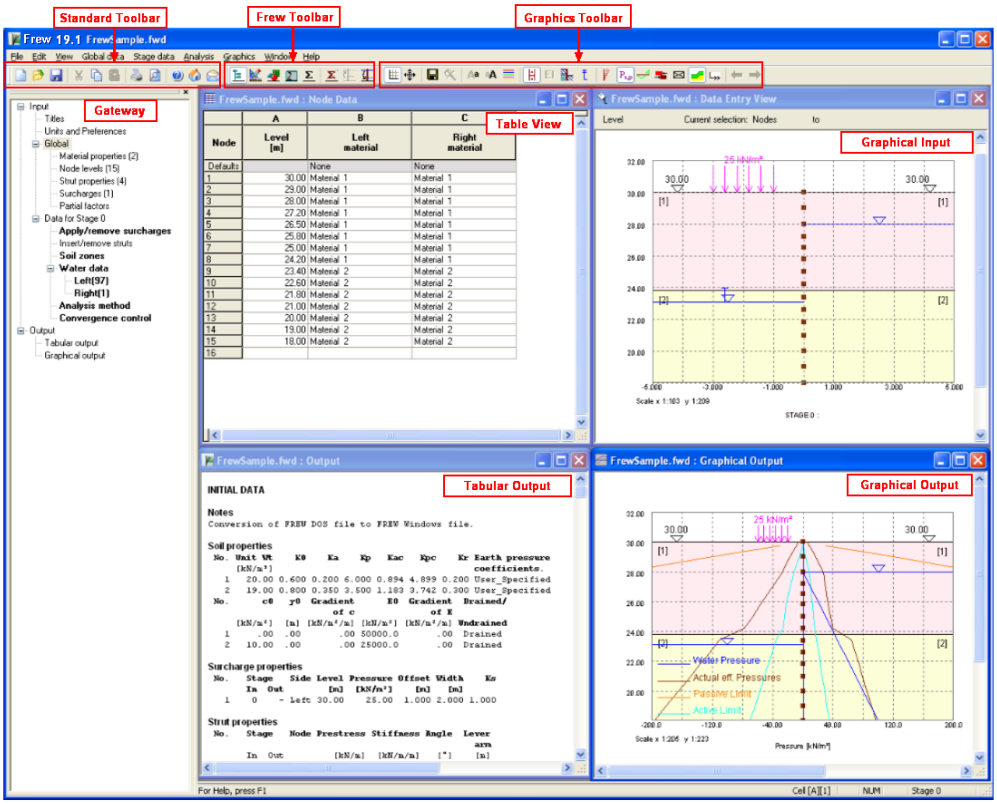
- All methods allow rigid (vertical) boundaries at specified distances from the wall. A rigid base is also assumed at the lowest node for the "**Safe**" and Mindlin methods.

© Oasys Ltd. 2014

- Soil pressure limits, active and passive, may be redistributed to allow for arching effects.
- Any vertical distribution of Young's modulus may be specified, and each model provides an approximate representation of this distribution. Alternatively the user may specify Young's modulus as either constant for the Mindlin model or linearly variable for the "**Safe**" method if desired.
- The effect of summer expansion and winter contraction of integral bridges can be assessed using the integral bridge feature.
- Where appropriate the effect of seismic ground movement can be assessed using the Wood's and Mononobe-Okabe methods.

1.3 Components of the User Interface

The principal components of Frew's user interface are the Gateway, Table Views, Graphical Output, Tabular Output, toolbars, menus and input dialogs. These are illustrated below.



1.3.1 Working with the Gateway

The Gateway gives access to all the data that is available for setting up a Frew model.

Top level categories can be expanded by clicking on the `+´ symbol beside the name or by double clicking on the name. Clicking on the `-´ symbol or double clicking on the name when expanded will

© Oasys Ltd. 2014

close up the item. A branch in the view is fully expanded when the items have no symbol beside them.

Double clicking on an item will open the appropriate table view or dialog for data input. The gateway displays data from the current stage under "Data for Stage ..." node. The data items which have changed from the previous stage are indicated by bold font.

2 Methods of Analysis

Frew is used to compute the behaviour of a retaining wall through a series of construction sequences.

Displacement calculations are complex and involve considerable approximations. It is essential therefore, that the user understands these approximations and considers their limitations before deciding which type of analysis is appropriate to the problem. The main features of the computations are summarised here. Further details are presented in the section on "Detailed Processes in Frew".

A summary of the **Frew** analysis, for inclusion with the program results and project reports, is included in Brief technical description.

2.1 Stability Check

The stability check calculations assume limit equilibrium, i.e. limiting active and passive states either side of the wall.

These pressures are used to calculate the required penetration of the wall to achieve rotational stability.

Support for partial factor analysis is now available in the program.The user may specify this in "Partial Factors" dialog.

Two statically determinate mechanisms in the form of "Fixed earth" cantilever and "Free earth" propped retaining walls can be solved. For either problem several struts with specified forces can be applied.

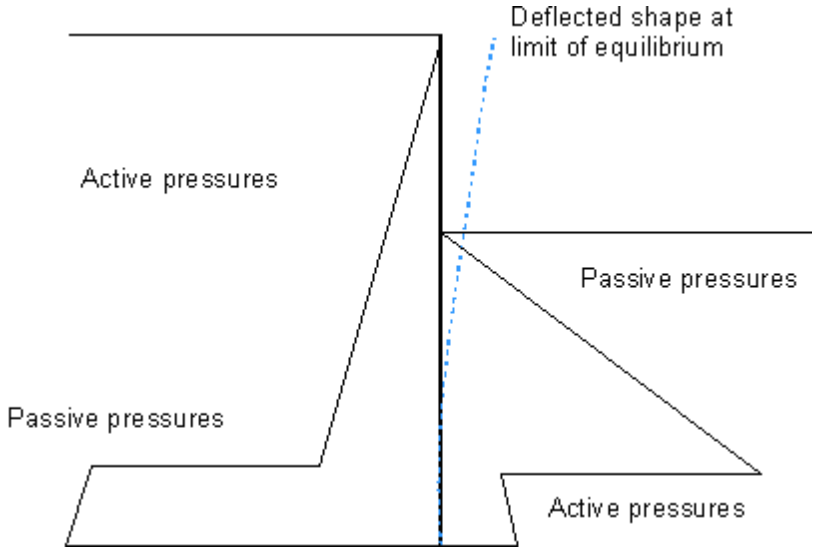
Note : The user should be aware that other mechanisms of collapse may exist for the problem which are not considered by the stability check. These include rotation of the soil mass, failure of the props/anchors or failure of the wall in bending.

2.1.1 Fixed Earth Mechanisms

This method is used to model cantilever walls.

The mechanism assumes that the wall is fixed by a passive force developing near its base. The level of the base of the wall is calculated to give equilibrium under this assumed pressure distribution.

© Oasys Ltd. 2014

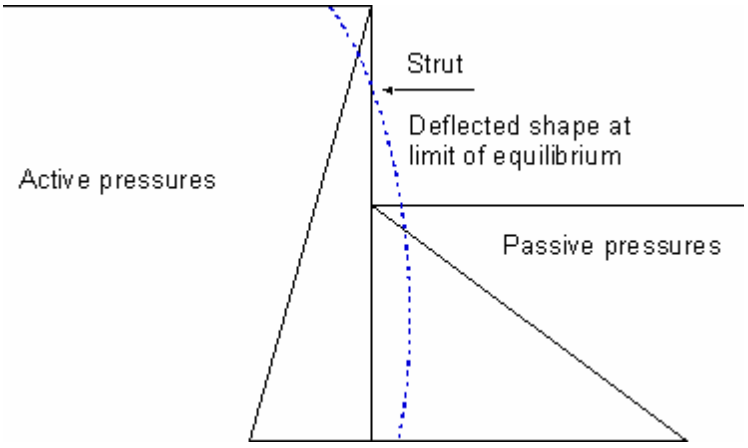


Pressure diagram for Fixed Earth mechanism

2.1.2 Free Earth Mechanisms

This method is used to model propped walls.

The mechanism assumes rotation about a specified strut and calculates the level of the base of the wall and the force in the strut required to give equilibrium.



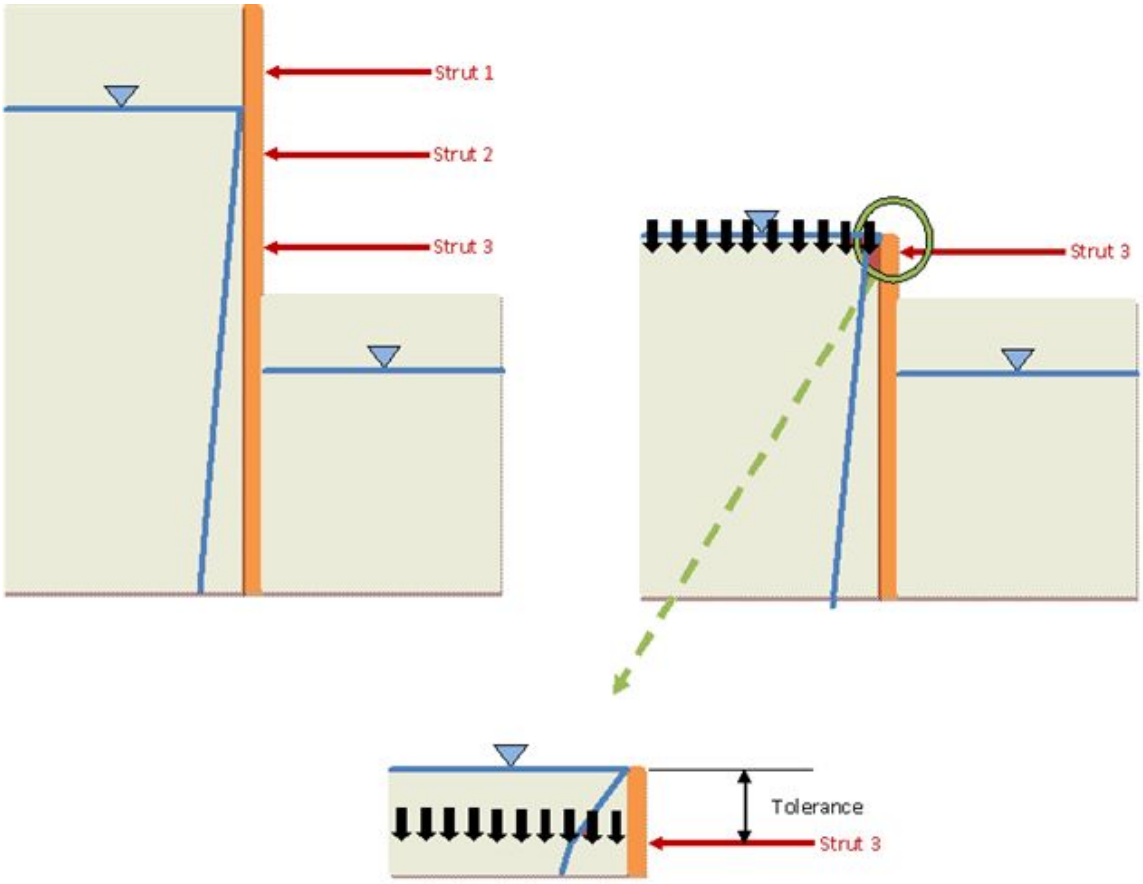
Pressure diagram for Free Earth mechanism

2.1.2.1 Multi-propped walls

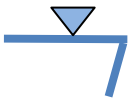
When the model has more than one active strut, the lowest strut is taken as the rotation strut by the program. The following assumptions and approximations are made:

- The ground level on the retained side is assumed to be 1 cm above the rotation strut.
- Any soil layers above this assumed ground level are treated as equivalent surcharges.
- The ground water distribution is also applied starting from the assumed ground level. However, the pore pressure from the level of rotation strut downwards are same as the original pore pressure distribution. The pore pressure from the level of assumed ground level to the level of rotation strut is assumed to vary linearly.

© Oasys Ltd. 2014



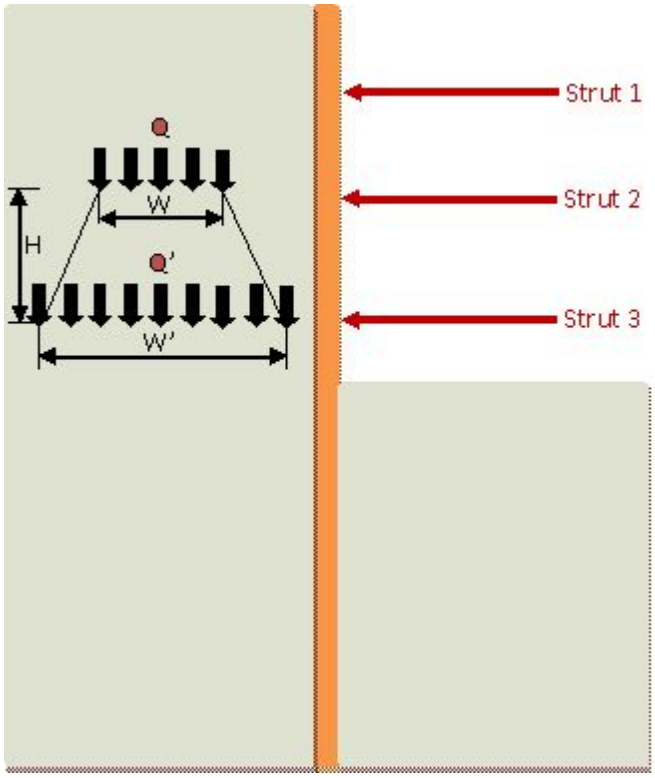
Equivalent surcharge from soil above the rotation strut.

 Pore pressure distribution

Tolerance is related to the assumed location of ground level above the location of lowest strut. This is currently taken as 1 cm.

- Any strip surcharges that are present above the location of rotation strut are modelled as equivalent strip surcharges at the level of the lowest strut. The load intensity and width of this equivalent surcharge are calculated using 2:1 rule for diffusion of vertical stress in soil.

© Oasys Ltd. 2014



In the example above,

Q' = W*Q/W'

W' = W + 1/2*H + 1/2*H = W + H

where,

W = width of the strip surcharge,

Q = load intensity of the strip surcharge in kN/m^2,

W' = width of equivalent surcharge,

Q' = load intensity of equivalent surcharge in kN/m^2,

H = height of the strip surcharge above the level of rotation strut.

Generally speaking, the centrelines of the actual surcharge and the equivalent surcharge coincide. However, if the extent of equivalent surcharge crosses the wall, then the equivalent surcharge is assumed to have the same width calculated as above, but it is assumed to start from the edge of the wall.

Note: The partial factors for user-defined surcharges are not applied to the equivalent surcharge due to overburden above the lowest strut in this analysis of multi-propped walls.

© Oasys Ltd. 2014

2.1.3 Active and Passive Limits

Active and passive pressures are calculated at the top and base level of each stratum and at intermediate levels. These are placed where there is a change in linear profile of pressure with depth.

The generation of intermediate levels ensures the accuracy of the calculation of bending moments and shear forces. Intermediate levels will be generated where there is a change in the linear profile of pressure with depth e.g.

- at water table levels/piezometric points
- at surcharge levels
- at intervals of 0.5 units within a stratum with a cohesion strength component

The effective active and passive pressures are denoted by p'_a and p'_p respectively. These are calculated from the following equations:-

$$p'_a = k_a \sigma'_v - k_{ac} c'$$

$$p'_p = k_p \sigma'_v + k_{pc} c'$$

where

c' = effective cohesion or undrained strength as appropriate

σ'_v = vertical effective overburden pressure

Note : Modification of the vertical effective stress due to wall friction should be made by taking appropriate values of k_a and k_p.

k_a and k_p =horizontal coefficients of active and passive pressure

k_{ac} and k_{pc} =cohesive coefficients of active and passive pressure

k_{ac} and k_{pc} can be evaluated as:

$$K_{ac} = 2\sqrt{K_a}\sqrt{\left\{1 + \frac{c_w}{c'}\right\}}$$

$$K_{pc} = 2\sqrt{K_p}\sqrt{\left\{1 + \frac{c_w}{c'}\right\}}$$

Where

c_w = wall adhesion

Note : For conditions of total stress $k_a = k_p = 1$.

© Oasys Ltd. 2014

For a given depth z

$$\sigma'_v = \int_0^z \gamma_s \, dz + \sigma_{zudl} - u$$

where

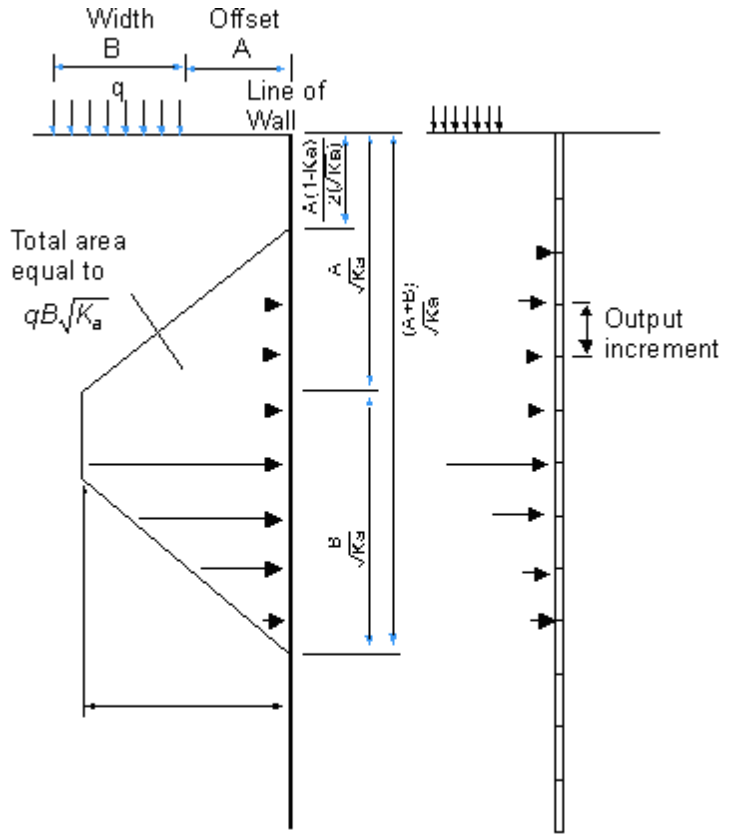
γ_s = unit weight of soil
u = pore water pressure
σ_{zudl} = vertical sum of pressures of all uniformly distributed loads (udl's) above depth z.

A minimum value of zero is assumed for the value of ($k_a\sigma'_v$ - $k_{ac}c'$).

Effect of strip surcharges

The effect on the active pressure of strip surcharges is calculated by the method of Pappin et al (1986), also reported in Institution of Structural Engineers (1986).

The approximation which has been derived is shown below :



Approximation to represent the change in active pressure due to a strip surcharge

Approximation of active pressure by equivalent forces (used in STAWAL)

© Oasys Ltd. 2014

Note : If the width of the load (B) is small, the diagram will become triangular.

The additional active pressure due to the surcharge is replaced by a series of equivalent forces. These act at the same spacing of the output increment down the wall. Thus a smaller output increment will increase the accuracy of the calculation.

Varying values of k_a

If the active pressure coefficient k_a varies with depth, the program chooses a mean value of k_a between any depth z and the level of the surcharge. **Stawal** then imposes the criterion that the active force due to the surcharge down to depth z be equal to the force derived from the diagram in above.

This is then subjected to the further limitation that the pressure does not exceed $qk_{az.}$

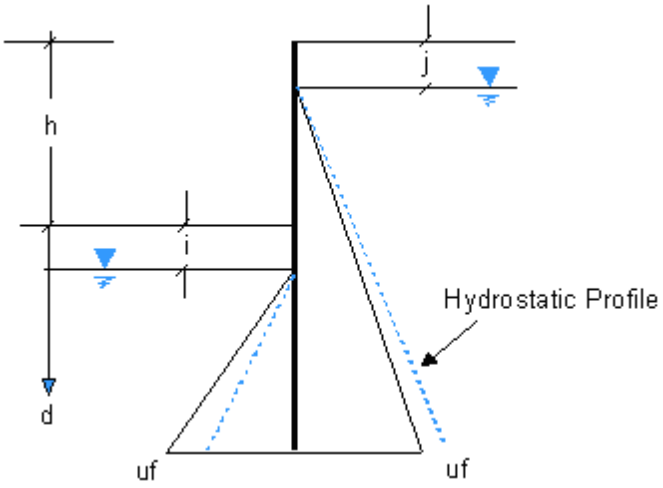
where

q = surcharge pressure.

k_{az} = active pressure coefficient to depth z.

2.1.4 Groundwater Flow

Water flow beneath the base of the wall can be modelled by setting the "Balance water pressures" switch in the "Stability Check" dialog box. The program uses the following iterative process



1. Carry out initial calculation using input water data to obtain the first estimate of embedment of the wall (d).

2. Calculate U_f for embedment d.

Alter the ground water gradient either side of the wall by specifying a piezometric pressure equivalent to U_f at the base of the wall, where γ_w =10.

© Oasys Ltd. 2014

3. Re-run the Stability analysis.

4. Check calculated value of d.

5. Repeat steps 2 to 4 until d is consistent with the groundwater profile and Uf is balanced at the base.

Note: This modification to water profile is only for stability calculations. It is **NOT** carried over to the actual Frew analysis.

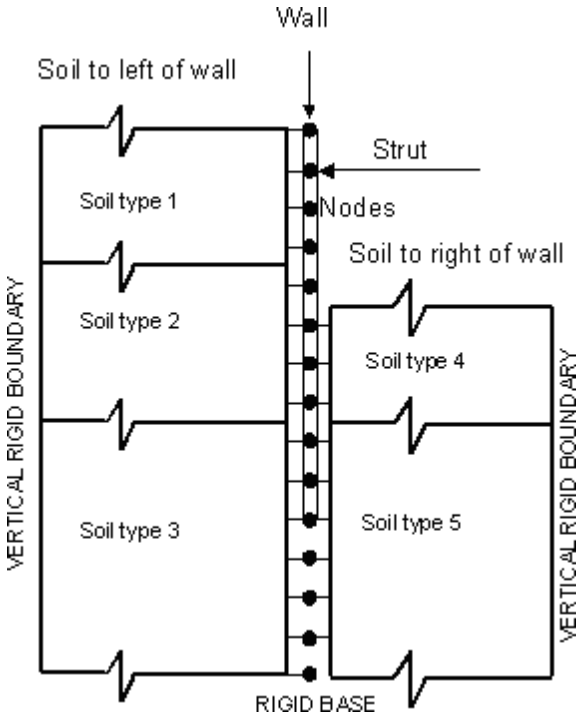
2.2 Full Analysis

The analysis is carried out in steps corresponding to the proposed stages of excavation and construction. An example, showing typical stages of construction that can be modelled, is given in Assembling Data.

The initial stage (Stage 0) is used to calculate the soil stress prior to the installation of the wall. Displacements computed in this stage are set to zero.

At each stage thereafter the incremental displacements, due to the changes caused by that stage, are calculated and added to the existing displacements. The soil stresses, strut forces, wall bending moments and shear forces are then determined.

The numerical representation is shown below.



The wall is modelled as a series of elastic beam elements joined at the nodes. The lowest node is either the base of the wall or at a prescribed rigid base in the ground beneath the wall.

© Oasys Ltd. 2014

The soil at each side of the wall is connected at the nodes as shown on the figure.

At each stage of construction the analysis comprises the following steps:

a) The initial earth pressures and the out of balance nodal forces are calculated assuming no movement of the nodes.

b) The stiffness matrices representing the soil on either side of the wall and the wall itself are assembled.

c) These matrices are combined, together with any stiffness' representing the actions of struts or anchors, to form an overall stiffness matrix.

d) The incremental nodal displacements are calculated from the nodal forces acting on the overall stiffness matrix assuming linear elastic behaviour.

e) The earth pressures at each node are calculated by adding the changes in earth pressure, due to the current stage, to the initial earth pressures. The derivation of the changes in earth pressure involves multiplying the incremental nodal displacements by the soil stiffness matrices.

f) The earth pressures are compared with soil strength limitation criteria; conventionally taken as either the active or passive limits. If any strength criterion is infringed a set of nodal correction forces is calculated. These forces are used to restore earth pressures, which are consistent with the strength criteria and also model the consequent plastic deformation within the soil.

g) A new set of nodal forces is calculated by adding the nodal correction forces to those calculated in step (a).

h) Steps (d) to (g) are repeated until convergence is achieved.

i) Total nodal displacements, earth pressures, strut forces and wall shear stresses and bending moments are calculated.

2.3 Soil Models

The soil, on both sides of the wall, is represented as a linear elastic material which is subject to active and passive limits. Three linear elastic soil models are available in **Frew**.

1. **Safe** Method.
2. Mindlin Method.
3. Sub-grade Reaction Model.

Note: The **sub-grade reaction model** is not currently active, it will be added to **Frew** in the near future.

All use different methods to represent the reaction of the soil in the **elastic phase**.

© Oasys Ltd. 2014

2.3.1 Safe Method

This method uses a pre-calculated soil stiffness matrix developed from the Oasys **Safe** program.

The soil is represented as an elastic continuum. It can be **'fixed'** to the wall, thereby representing full friction between the soil and wall. Alternatively the soil can be **'free'**, assuming no soil/wall friction, see Fixed or Free solution.

Accuracy of the Safe solution

This method interpolates from previously calculated and saved results, using finite element analysis from the **Safe** program.

The method gives good approximations for plane strain situations where Young's modulus is constant or increases linearly from zero at the free surface.

For a linear increase in Young's modulus from non-zero at the free surface the results are also good, but for more complicated variations in layered materials the approximations become less reliable.

In many situations when props or struts are being used, "fixed" and "free" give similar results. An exception is a cantilever situation where the "fixed" method will give less displacements because it models greater fixity between the soil and wall.

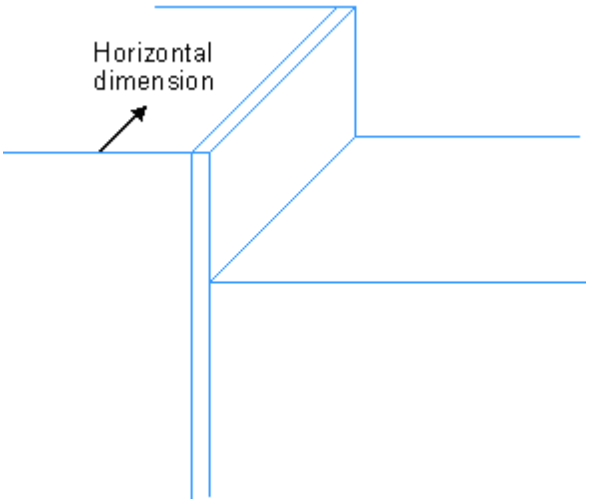
It must be noted that the case with interface friction ("**fixed**") is somewhat approximate because Poisson's ratio effects are not well modelled. For example, these effects in a complete elastic solution can **cause outward movement of the wall when there is a shallow soil excavation**.

For detailed information on the approximations and thereby the accuracy of the **Safe** method see Approximations used in the Safe Method.

2.3.2 Mindlin Method

The Mindlin method represents the soil as an elastic continuum modelled by integrated forms of Mindlin's elasticity equations (Vaziri et al 1982). The advantage of this method is that a wall of finite length in the third (horizontal) dimension may be approximately modelled. It also assumes that the soil/wall interface has no friction.

© Oasys Ltd. 2014



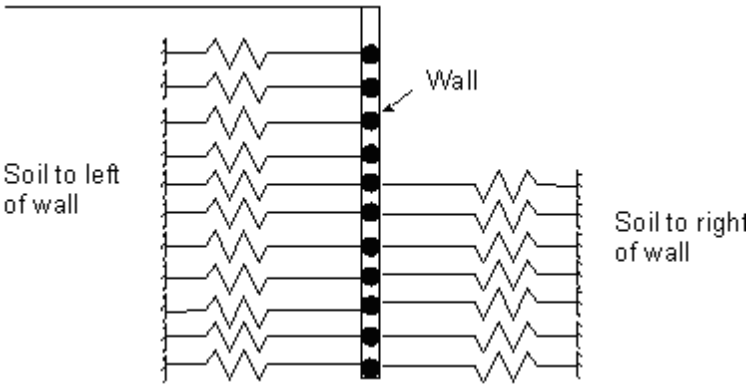
Accuracy of the Mindlin Solution

The method is only strictly accurate for a soil with a constant Young's modulus. Approximations are adopted for variable modulus with depth and as with the "**Safe**" method the user can override this by setting a constant modulus value.

For further information, see Approximations used in the Mindlin method.

2.3.3 Method of Sub-grade Reaction

The soil may be represented by a Sub-grade Reaction model consisting of non-interacting springs.



The stiffness is computed as:

$$K = EA / L$$

where

E = Young's modulus of the soil

A = distance between the mid-point of the elements immediately above and below the node under consideration

© Oasys Ltd. 2014

L = spring length (input by the user)

It is considered that this model is not realistic for most retaining walls, and no assistance can be given here for the choice of spring length, which affects the spring stiffness.

Note: The **sub-grade reaction model** is not currently active, it will be added to **Frew** in the near future.

2.4 Active and Passive Pressures

The active and passive pressures are calculated from the following equations:
Note: The brackets [ ] indicate the active pressure is only applied when the active force (from the surface to level z) is positive. Otherwise the pressure is set to zero.

$$p_a = [k_a \sigma'_v - k_{ac} c] + u$$
$$p_p = k_p \sigma'_v + k_{pc} c + u$$

Where

p_a and p_p	= active and passive pressures
k_a and k_p	= coefficients of active and passive pressure
k_{ac} and k_{pc}	= coefficients of active and passive pressure,
c	= effective cohesion or undrained shear strength as appropriate
u	= prescribed pore pressure

with soil cohesion these are generally set to

$$K_{ac} = 2\sqrt{K_a}\sqrt{\left\{1 + \frac{c_w}{c}\right\}}$$

$$K_{pc} = 2\sqrt{K_p}\sqrt{\left\{1 + \frac{c_w}{c}\right\}}$$

Where

c_w = cohesion between wall and soil

σ'_v = vertical effective stress

Wall friction should be allowed for in selecting values of k_a and k_p. Undrained behaviour can be represented by setting k_a and k_p to unity with appropriate values of c, k_{ac} and k_{pc}.

The use of **redistribution** can allow for the effects of arching in the soil.

If "no redistribution" is specified, the wall pressures at all points are limited to lie between p_a and p_p. However, if "redistribution" is allowed, it is assumed that arching may take place according to theory presented in Calculation of Active and Passive Limits and Application of Redistribution.

Note: It is considered that the "redistribution" option, while still being somewhat conservative,

© Oasys Ltd. 2014

represents the "real" behaviour much more accurately.

If **surcharges** of limited extent are specified above the level in question the active pressure is increased in accordance with the theory presented in Active Pressures due to Strip Load Surcharges.

However, strip surcharges **are not included** in calculating passive pressure (see Passive Pressures due to Strip Load Surcharges).

2.4.1 Effects of Excavation and Backfill

Excavation, backfill or changes of pore pressure cause a change in vertical effective stress $\Delta\sigma'_v$. It is assumed that, **in the absence of wall movement**, the change in horizontal effective stress $\Delta\sigma'_h$ will be given by

$$\Delta\sigma'_h = K_r \Delta\sigma'_v$$

For an isotropic elastic material:

$$K_r = \nu / (1 - \nu)$$

where

ν = Poisson's ratio for drained behaviour

For **drained** behaviour, the typical range of K_r would be 0.1 to 0.5.

For **undrained** behaviour, the same approach is applied to total stress. In this case, the undrained Poisson's ratio would normally be taken to be 0.5, where $K_r = 1.0$

When filling, the horizontal effective stresses **in the fill material** are initially set to K_0 times the vertical effective stress.

i.e. $\sigma'_h = K_0 \sigma'_v$

2.4.2 Calculation of Earth Pressure Coefficients

The equations presented below are taken from EC7 (1995) Annex G. They have been simplified to account only for vertical walls, with a vertical surcharge on the retained side. The following symbols are used in the equations:

ϕ' angle of shearing resistance of soil (degrees)

δ wall/soil friction angle (degrees)

β angle of ground surface to horizontal (degrees)

The coefficient of **horizontal** earth pressure, K_h is given by:

© Oasys Ltd. 2014

where,

$$K_h = \cos^2 \beta . \left(\frac{1 + \sin\phi' \sin(2m_w + \phi')}{1 - \sin\phi' \sin(2m_t + \phi')} \right) . \exp(2v \tan\phi')$$

$$m_t = \frac{\cos^{-1}\left(-\frac{\sin\beta}{\sin\phi'}\right) - \phi' - \beta}{2}$$

$$m_w = \frac{\cos^{-1}\left(\frac{\sin\delta}{\sin\phi'}\right) - \phi' - \delta}{2}.$$

And

$$v = m_t + \beta - m_w$$

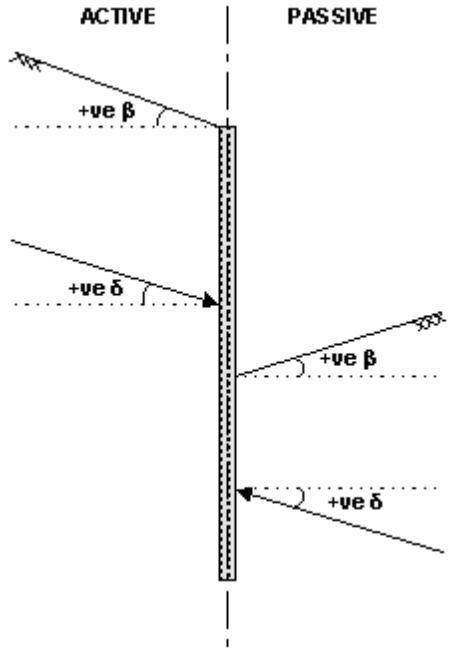
m_t , m_w and v have units of degrees. However, v must be converted into **radians** before substitution into the above equation for evaluating K_h.

For calculation of **active** earth pressure coefficients, the angle of shearing resistance of the soil and the wall/soil friction angle must be entered as **negative** values.

For calculation of **passive** earth pressure coefficients **positive** angles should be used.

For both active and passive earth pressure coefficients the value of β is **positive** for a ground level which **increases** with distance from the wall.

© Oasys Ltd. 2014



2.5 Total and Effective Stress

Frew recognises two components of pressure acting on each side of the wall.

1. Pore pressure (u). This is prescribed by the user and is independent of movement.
2. Effective stress (Pe). This has initial values determined by multiplying the vertical effective stress by the coefficient of earth pressure at rest (K_0).

Thereafter its values change in response to excavation, filling and wall movement.

The vertical effective stress is calculated as:

$$\sigma_v^{'} = \int_z^{z_s} \gamma dz - u + \sigma_{zudl}$$

where:

u = Prescribed pore pressure
g = unit weight
z = level
z_s = surface level
σ_{zudl} = vertical stress due to all uniformly distributed surcharges above level z.

© Oasys Ltd. 2014

2.5.1 Drained Materials

Effective stress and pore pressure are used directly to represent drained behaviour.

Note: The pore pressure profile is defined by the user and is independent of movement.

2.5.2 Undrained Materials and Calculated Pore Pressures

Frew can be requested to calculate undrained pore pressures at each stage.

The feature is available in the Material Properties table, and is activated by specifying for an undrained material another material zone from which effective stress parameters are to be taken. A "shape factor" is also required, which controls the shape of the permitted effective stress path for undrained behaviour. The default value for the shape factor is 1, which prevents occurrence of any effective stress state outside the Mohr-Coulomb failure envelope, but it can optionally be revised to 0, representing a Modified Cam-Clay envelope, or any value in between. [NB: values less than 1 have not been validated and use of a value less than 1 is not recommended. The option is retained in the program for experimental purposes. A spreadsheet 'undr_dr_calc.xls' is provided in the 'Samples' sub-folder of the program installation folder. This allows the user to experiment with values for the various parameters and with the shape factor, if wished.]

If reasonable values of pore pressure are not used during undrained behaviour, then on transition to drained behaviour, the program may not calculate displacements with satisfactory accuracy. The calculation in **Frew** aims to provide a reasonable set of undrained pore pressures. Given the relative simplicity of **Frew** and the present state of knowledge of soil behaviour, they cannot be accurate (although should be better than a user-defined pore pressure profile) and the user should check that they appear reasonable. Guidance on warning messages is given below.

The process used in the program can be understood by studying the stress path plot below. Failure in an undrained material occurs at the intersection of the ϕ' and C_u lines. This point is derived from the effective stress parameters of the "material number for effective stress parameters" specified by the user for each undrained material. The envelope of possible total stress values is shown in red (examples for shape factors of 1 and 0.75 are shown); this is taken to be elliptical except where reduced by shape factors > 0. The calculated undrained effective stress path is shown in blue.

For each iteration in an undrained stage, the program calculates the total stress and the effective stress, using the value on the blue effective stress path unless limited by the red envelope. The undrained pore pressure is then the difference between the total and effective stresses.

The diagram shows that, if the shape factor is less than 1, it is possible for the effective stress calculated to lie outside the limits of the effective stress parameters; this would lead to some changes in total stress, and hence displacement, in the transition from undrained to effective stress behaviour. This problem is avoided by following the recommendation to use the default shape factor of 1.0.

Advice on "data" pore pressures when using this feature

Any pore pressures entered by the user will be ignored in an undrained material for which automatic calculation of pore pressures has been requested (i.e. by setting a valid "material for effective stress parameters" in the Materials table).

© Oasys Ltd. 2014

Warning messages

These appear as symbols in the node results tables for any stage which calculates undrained pore pressures, and a brief explanation is added in a footnote to the table.

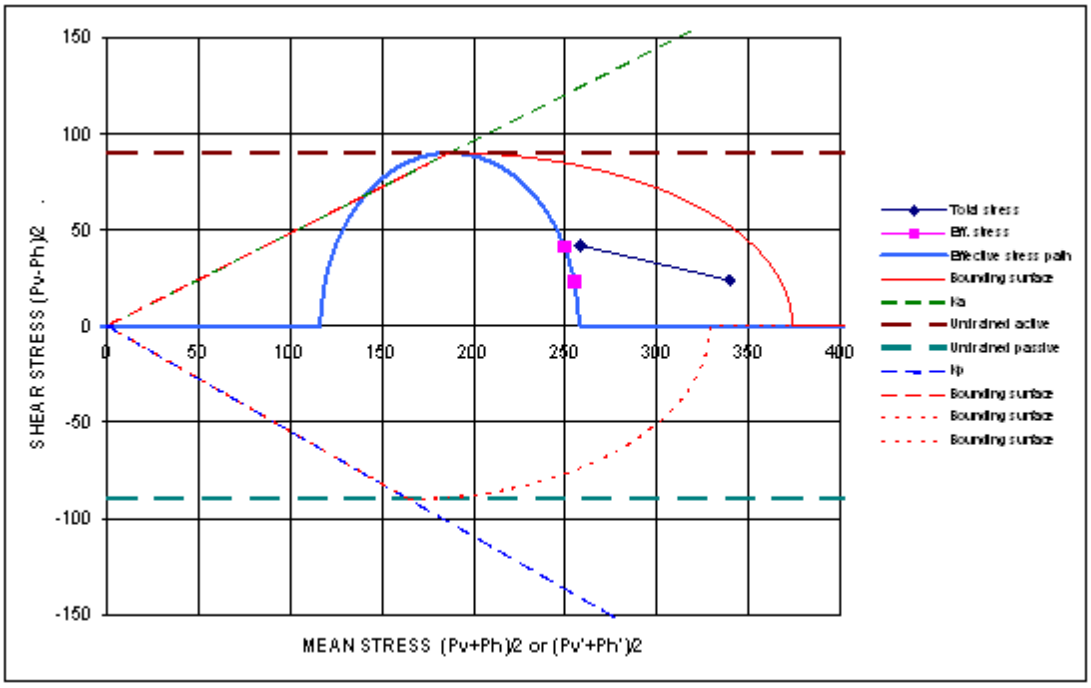
's' initial stress outside effective strength limits

This situation should not occur and probably reflects a data error in which either the user has change the effective stress parameters between stages, or, in the first stage, inconsistent values of K_o, K_a and K_p have been specified.

It could possibly be detected on returning to use of effective stress parameters after an undrained stage with FACTOR < 1.

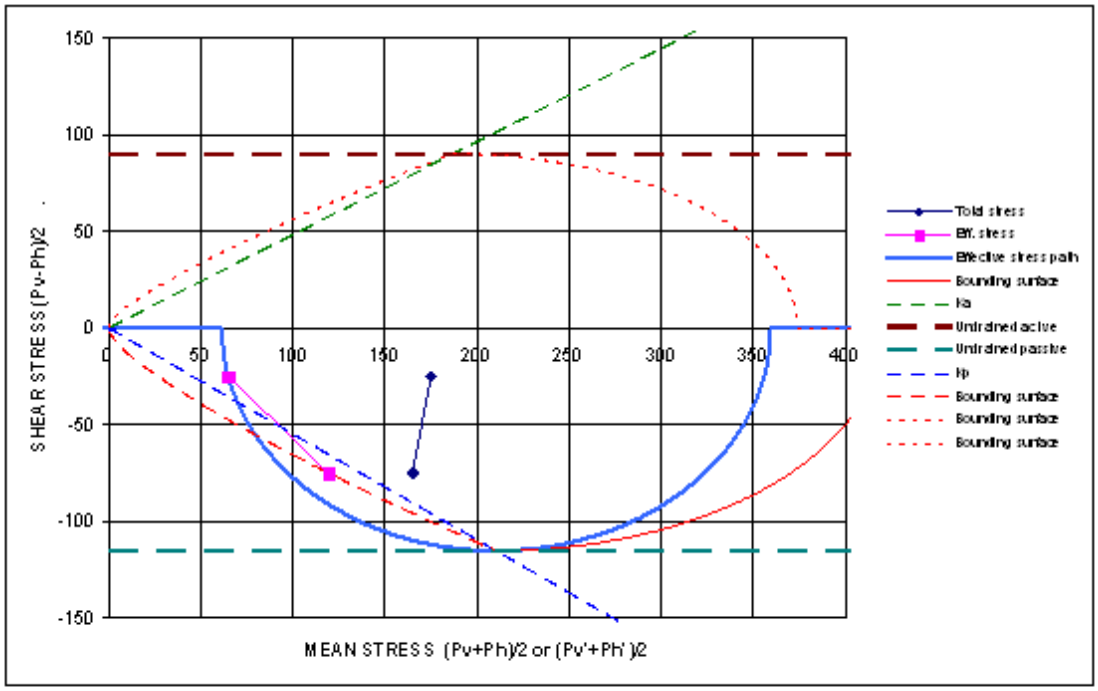
'u' undrained strength unreasonably low for stress state

At an earlier drained stage, probably at initialisation, the program has calculated a horizontal stress which exceeds the undrained strength limits specified by the user, in relation to the vertical stress at the node. This may be due to incorrect data, i.e undrained strength not increasing in a sensible manner with depth, or too low a value of constant C_u .



Factor = 1

© Oasys Ltd. 2014



Factor = 0.75

2.5.3 Undrained Materials and User-defined Pore Pressure

In this procedure, σ_v and σ_h are 'real' values which are used in the calculations for equilibrium; whilst u may be a user-specified value which does not change during deformation, and is therefore not generally correct for true undrained behaviour. Hence the results reported as V_e (Vertical stresses) and P_e (horizontal stresses) are **apparent** rather than true effective stresses, unless the program calculates approximate undrained pore pressures.

For undrained or partially drained behaviour (where pore pressures change in response to movements), a constant pore pressure component (u_0) defined by the user, is thereby very unlikely to represent the actual pore pressure in the soil. Approximate undrained pore pressures can be calculated by the program by setting an extra material parameter, see Undrained Materials and Calculated Pore Pressures . If this option is selected, any "data" pore pressure distribution entered by the user is ignored in undrained materials.

There are two ways of representing undrained materials, if undrained pore pressures are not being calculated:

A. Specified profile of pore water pressure

- Here, the pore pressure considered by the program can usually be regarded as the initial pore pressure before deformation, whilst
- The apparent horizontal effective stress (P_e) becomes the sum of the true effective earth pressures and the excess pore pressures due to deformation.

$$P_e = \sigma'_h + \Delta u$$

© Oasys Ltd. 2014

B. Zero pore water pressure profile

- Here, the value of apparent horizontal effective stress (P_e) can then be equated to the true total stress.

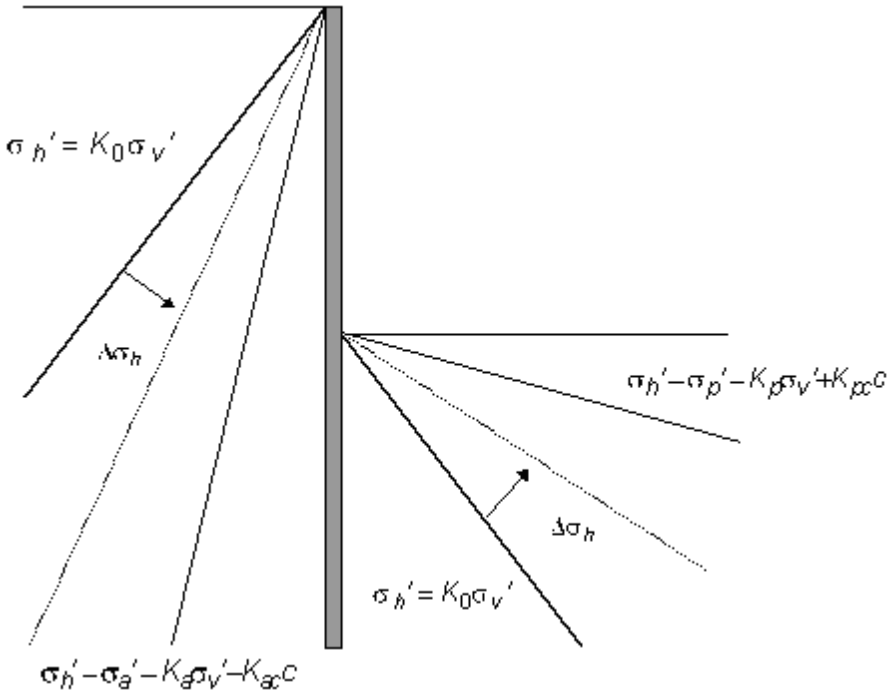
$$P_e = \sigma'_h + u$$

In both these cases the values of K_a, K_p and K_r should be set to unity (1.0), but with non-zero undrained strengths (c) and coefficients K_{ac} and K_{pc}.

Calculation procedure

Frew executes the following calculation procedure. This includes for a profile of pore pressures, if specified, as indicated.

1. Calculation of the total vertical stress σ_v.
2. Calculation of the effective vertical stress where:
   $\sigma'_v = \sigma_v - u$
3. Checks to make sure that $\sigma_v \geq 0$. The program stops and provides an error message if this is not so.
4. Calculation of the minimum active effective stress:
   $\sigma'_a = K_a\sigma'_v - K_{ac}c$
5. Checks to make sure that $\sigma'a \geq 0$ (i.e. $K_a\sigma'_v \geq K_{ac}c$). If the value is less than zero then the program resets to 0. So generally:
   $\sigma'_a = K_a(\sigma_v - u) - K_{ac}c \geq 0$



© Oasys Ltd. 2014

Note: **Frew** uses σ'_a as a lower limit on the horizontal effective stress σ'_h. σ'_h is used in the equilibrium equations, for determination of the wall deflection, where $\sigma'_h = \sigma_h + u$

Effect of specified pore pressure

Specified pore pressure, (u) takes effect in steps 2 and 5.
Step 2 - Specified pore water pressure reduces the effective vertical stress.
Step 5 - Limits the apparent horizontal effective stresses to ≥ 0.
In earlier versions of **Frew**, this could be used to advantage because the specified 'pore pressure' u could become equivalent to a 'minimum fluid pressure'. There is now a completely separate feature in which a minimum equivalent fluid pressure (MEFP) can be specified, see Minimum Equivalent Fluid Pressure .

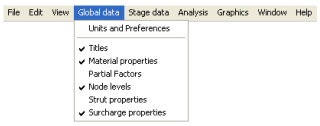
2.5.4 Undrained to Drained Example

An example file (Undr-PP-Example.fwd) is available in the Samples sub-folder of the program installation folder. The user can see from this the way that the feature for automatic calculation of undrained pore pressures has been used.

3 Input Data

Data is input via the Global Data and Stage Data menus, or via the Gateway. Some basic and global data can be input to a new file using the New Model Wizard, but the following gives some background on the way the data is organised and can be edited after initial entry.

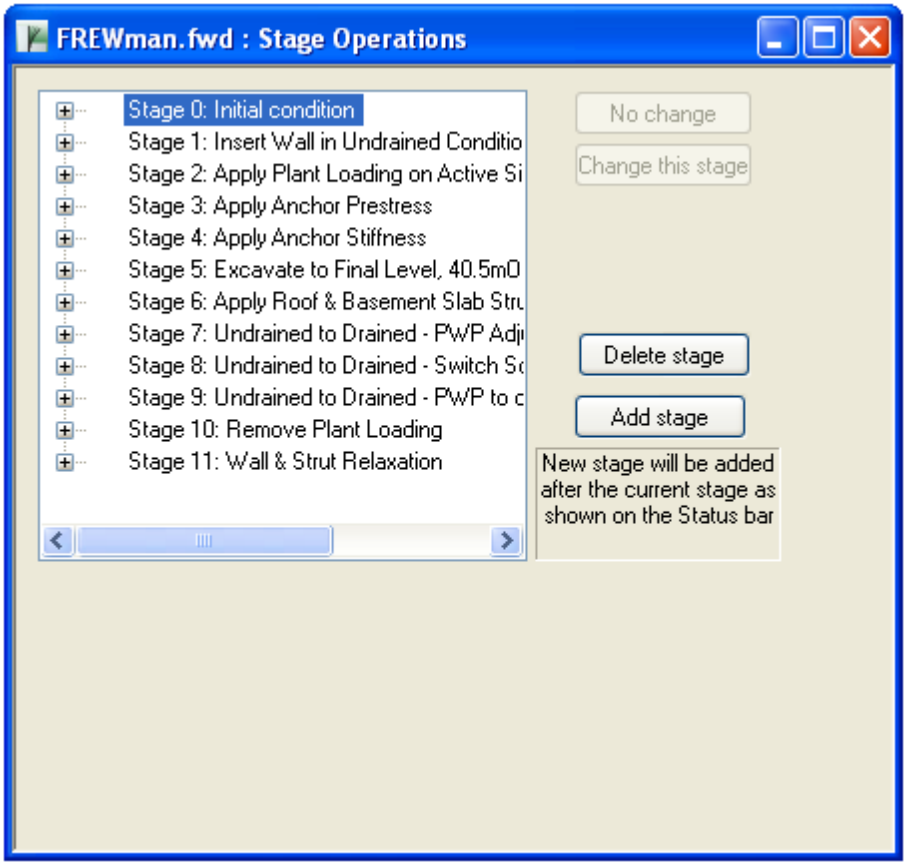
Global Data



The **Global Data** menu enables entry of the general data which is common to or accessed throughout the analysis. The information can be entered in any order. The exception is that the program requires Material properties to be entered before Node levels.

Stage Data

© Oasys Ltd. 2014


FREWman.fwd : Stage Operations
Stage 0: Initial condition
Stage 1: Insert Wall in Undrained Conditio
Stage 2: Apply Plant Loading on Active Si
Stage 3: Apply Anchor Prestress
Stage 4: Apply Anchor Stiffness
Stage 5: Excavate to Final Level, 40.5mO
Stage 6: Apply Roof & Basement Slab Stru
Stage 7: Undrained to Drained - PWP Adj
Stage 8: Undrained to Drained - Switch Sc
Stage 9: Undrained to Drained - PWP to c
Stage 10: Remove Plant Loading
Stage 11: Wall & Strut Relaxation
No change
Change this stage
Delete stage
Add stage
New stage will be added after the current stage as shown on the Status bar

The **Stage Operations** window or the icon will allow individual stages to be modified. When opened, the Stage Operations view shows a tree diagram, which allows access to all available options for each stage. Ticks are placed against those options which have been changed.

This window also allows the creation of new stages of analysis and the deletion of stages that are no longer required.

Note: Left click on the boxes and to open or close the tree diagram for each stage.

Accessing data using Gateway:

The user can also access the "Global Data" menu items and the current stage menu items using the Gateway.

Whenever the data item in the current stage item is different from the previous stage, it is shown in bold.

3.1 Assembling Data

Each problem should be sub-divided into a series of **construction Stages** commencing with the initial stage, referred to by **Frew** as Stage 0. This stage defines the situation prior to the wall being installed in the ground and is therefore specified in terms of drained parameters. Various operations can be performed in subsequent stages, including changes from drained to undrained and vice versa.

© Oasys Ltd. 2014

Sketches showing the wall, soil strata, surcharges, water pressure, strut and excavation levels should be prepared for **each Stage**.

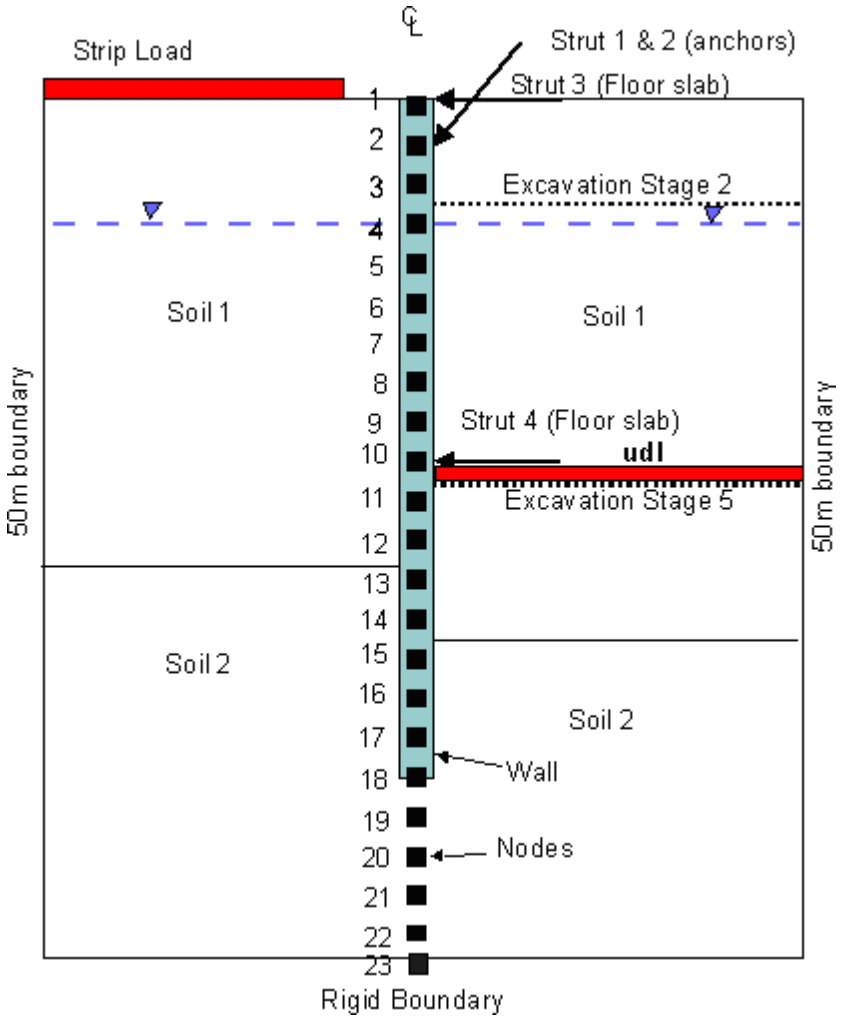
Examples of potential changes that can be applied during the construction stages are:

Stage 0	Set up initial stresses in the soil by adding the material types, groundwater conditions and applying any surcharges required prior to installing the wall. All materials should be set to **drained parameters** for this stage.
Stage 1	Install wall.
	Change to undrained materials (if required). In this example, undrained pore pressures are calculated by the program, see Undrained Materials and Calculated Pore Pressures.
Subsequent stages of construction	Excavate / backfill.
	Insert / Remove struts.
	Insert / Remove surcharges.
Long term effects	Return to drained parameters.
	Change groundwater conditions.
	Use the relaxation option to model the long term stiffness of the wall.

To illustrate these operations a manual example is given below.

General layout of manual example

© Oasys Ltd. 2014

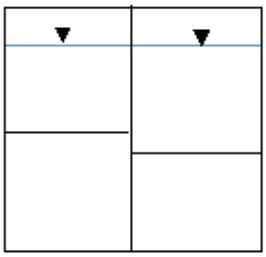


Combined stages shown here to aid placement of the nodes, see Nodes.

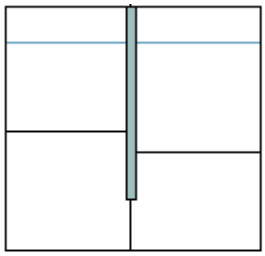
The following shows the construction sequence separated into stages ready for modelling.

Note: Soils 1 and 3 are Clay and have been used to represent the modelling of undrained material and the change to drained for long term conditions. **Soil 2** is a sand and thereby fully drained throughout the construction sequence.

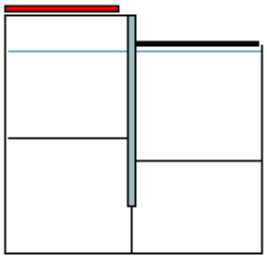
© Oasys Ltd. 2014



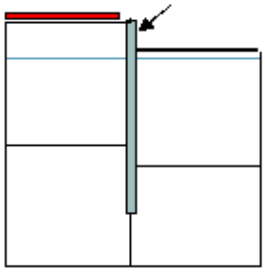
Stage 0 : Initial condition - drained



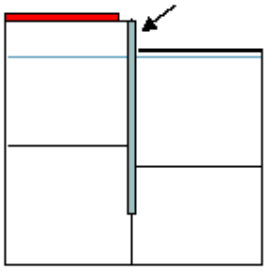
Stage 1 : Insert wall in undrained conditions



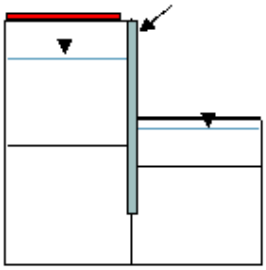
Stage 2 : Apply plant loading on active side and excavate to 47.5m OD



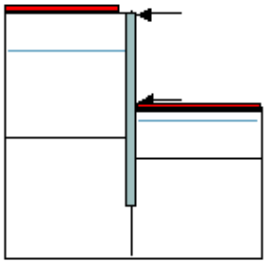
Stage 3 : Apply anchor prestress



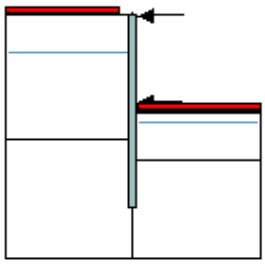
Stage 4 : Apply anchor stiffness



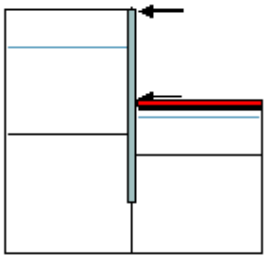
Stage 5 : Excavate to final level of 40.5m OD and dewater



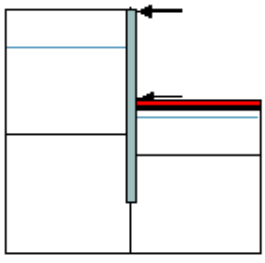
Stage 6 : Apply roof and basement slab struts and remove ground anchor



Stage 7 : Undrained to drained



Stage 8 : Remove plant loading



Stage 9 : Wall and strut relaxation

© Oasys Ltd. 2014

The program recalculates the displacements and forces within the system at each stage.

Several activities can be included within a single stage provided their effects are cumulative. For example it is appropriate to insert a strut and then excavate below the level of the strut in one stage, but it is not correct to excavate and then insert a strut at the base of the excavation in one stage. If in doubt the user should incorporate extra stages.

The computer model of the program geometry should be drawn with the wall node locations carefully selected in accordance with the guidance given in inserting Nodes.

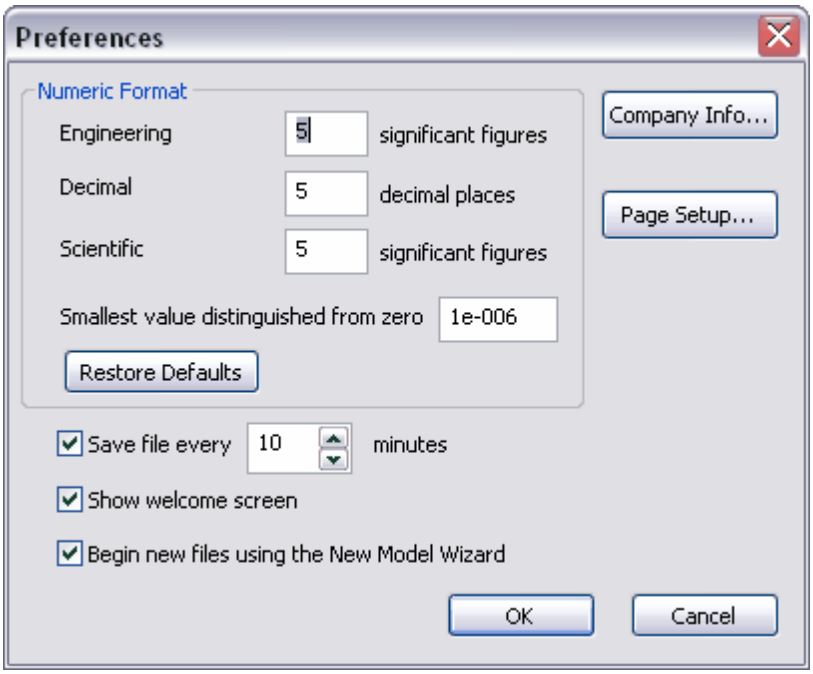
The nature of each problem will vary considerably and thereby the amount of data changes required for each construction stage. Some information is compulsory for the initial stages. Thereafter full flexibility is allowed in order to build up the correct progression of construction stages and long term effects.

	Stage 0 & Global data	**Stage 1**	**Construction stages**	**Long term**
Compulsory	Material properties (All materials) Node levels Soil zones (drained materials) Analysis Method Convergence control parameters	Wall properties		
Optional	Surcharges Struts Water	Analysis Method Convergence control parameters Soil zones (undrained or drained materials) Excavation or filling Surcharges Struts Water	Analysis Method Wall properties Convergence control parameters Soil zones (undrained or drained materials) Excavation or filling Surcharges Struts Water	Wall relaxation Analysis Method Convergence control parameters Soil zones (drained materials) Surcharges Struts Water

© Oasys Ltd. 2014

3.2 Preferences

The Preferences dialog is accessible by choosing Tools | Preferences from the program's menu. It allows the user to specify the units for entering the data and reporting the results of the calculations. These choices are stored in the computer's registry and are therefore associated with the program rather than the data file. All data files will adopt the same choices.



Numeric Format controls the output of numerical data in the Tabular Output. The Tabular Output presents input data and results in a variety of numeric formats, the format being selected to suit the data. Engineering, Decimal, and Scientific formats are supported. The numbers of significant figures or decimal places, and the smallest value distinguished from zero, may be set here by the user.

Restore Defaults resets the Numeric Format specifications to program defaults.

A time interval may be set to save data files automatically. Automatic saving can be disabled if required by clearing the "**Save file.**." check box.

Show Welcome Screen enables or disables the display of the Welcome Screen. The Welcome Screen will appear on program start-up, and give the option for the user to create a new file, to open an existing file by browsing, or to open a recently used file.

Begin new files using the New Model Wizard, if ticked, will lead the user through a series of screens to enter basic data for a new file. For more details, see New Model Wizard.

Company Info allows the user to change the company name and logo on the top of each page of print out. To add a bitmap enter the full path of the file. The bitmap will appear fitted into a space approximately 4cm by 1cm. The aspect ratio will be maintained. For internal Arup versions of the program the bitmap option is not available.

Page Setup opens a dialog which allows the user to specify the calculation sheet style for graphical and text printing e.g. whether it has borders and a company logo.

© Oasys Ltd. 2014

3.3 New Model Wizard

The New Model Wizard is accessed by selecting the `File | New´(Ctrl+N) option from the main menu, or by clicking the 'New' button on the Frew toolbar.

The New Model Wizard is designed to ensure that some basic settings and global data can be easily entered. It does not create an entire data file, and strut, surcharge and stage data should be entered once the wizard is complete.

Cancelling at any time will result in an empty document.

Note! The New Model Wizard can only be accessed if the "Begin new files using New Model Wizard" check box in Tools | Preferences is checked.

3.3.1 New Model Wizard: Titles and Units

The first property page of the New Model Wizard is the Titles and Units window. The following fields are available:

Job Number	allows entry of an identifying job number. The user can view previously used job numbers by clicking the drop-down button.
Initials	for entry of the users initials.
Date	this field is set by the program at the date the file is saved.
Job Title	allows a single line for entry of the job title.
Subtitle	allows a single line of additional job or calculation information.
Calculation Heading	allows a single line for the main calculation heading.

The titles are reproduced in the title block at the head of all printed information for the calculations. The fields should therefore be used to provide as many details as possible to identify the individual calculation runs.

An additional field for **notes** has also been included to allow the entry of a detailed description of the calculation. This can be reproduced at the start of the data output by selection of notes using File | Print Selection.

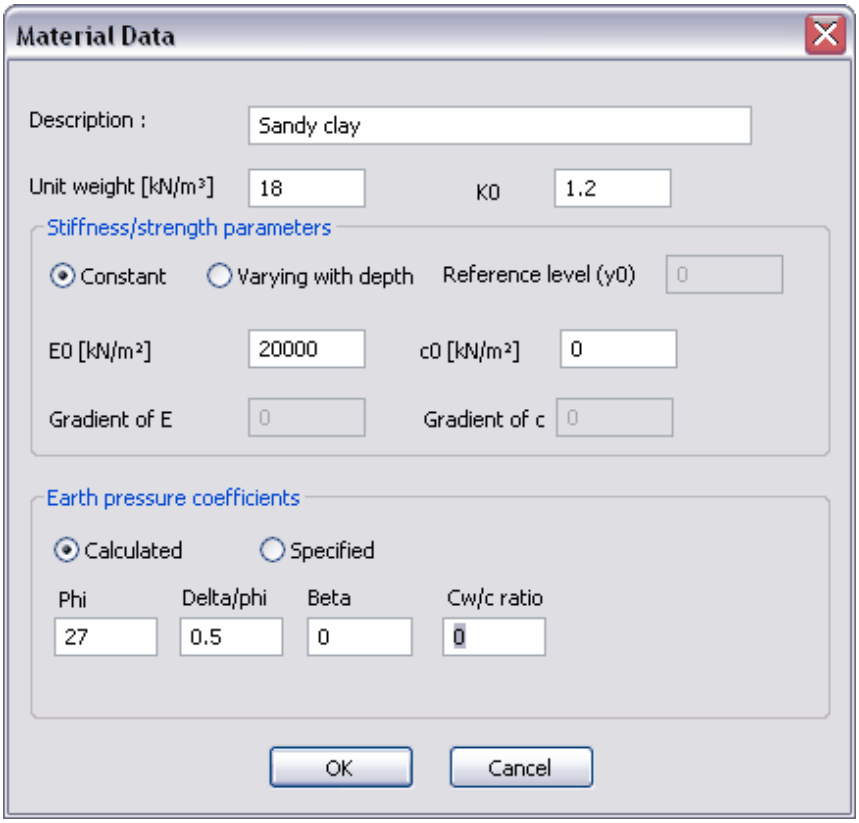
Clicking the Units button opens the standard units dialog.

3.3.2 New Model Wizard: Basic Data

The second wizard page contains the following options:

Problem geometry	Enter the levels of the top node and the lower rigid boundary. This will set the correct view range in subsequent graphical display.
Materials	Add or delete materials. Clicking "Add material" opens a further dialog allowing input of basic material data.

© Oasys Ltd. 2014



This data entry method will be sufficient in many cases, but some other settings (for example, undrained pore pressure calculation parameters) need to be set later in the normal Materials table.

Node generation

"Automatic" will allow all data input to be specified by level and node positions are generated by Frew.
"Manual" means that node positions must be entered by the user and most other data must be specified by node number rather than level.

Wall toe level

Selecting "Obtain from stability check" will enable the user to run a stability check before full analysis, to estimate the required toe level. This can be manually overridden if required.

To enter a known required toe level, select "Enter manually" and enter the level.

3.3.3 New Model Wizard: Stage Defaults

The final wizard page reproduces most of the Analysis Data dialog, and the values entered for analysis method, wall/soil interface, lateral boundary distances and Young's modulus specification will be used in generation of all new stages.

Clicking "Finish" completes the wizard and creates Stage 0 with the input data. The graphical input view will open to allow entry of node levels (if these are being created manually). If automatic node generation was selected, the graphical input view will show a single soil zone extending the full depth of the problem. More soil zones can be added as required to set up the initial ground profile

© Oasys Ltd. 2014

for Stage 0.

Strut and surcharge data is added separately, and additional stages created with the required stage changes, before proceeding to run a stability check and full analysis.

3.4 Global Data

Global data can be accessed from the Global data menu or the Gateway. The global data describes the problem as a whole. All the material properties, struts and surcharges which will be required for all subsequent stages must be defined here. If using the Automatic Node Generation feature, node levels are not required. If using manual node entry, the nodes must be placed in the correct locations to allow all subsequent construction stages to take place.

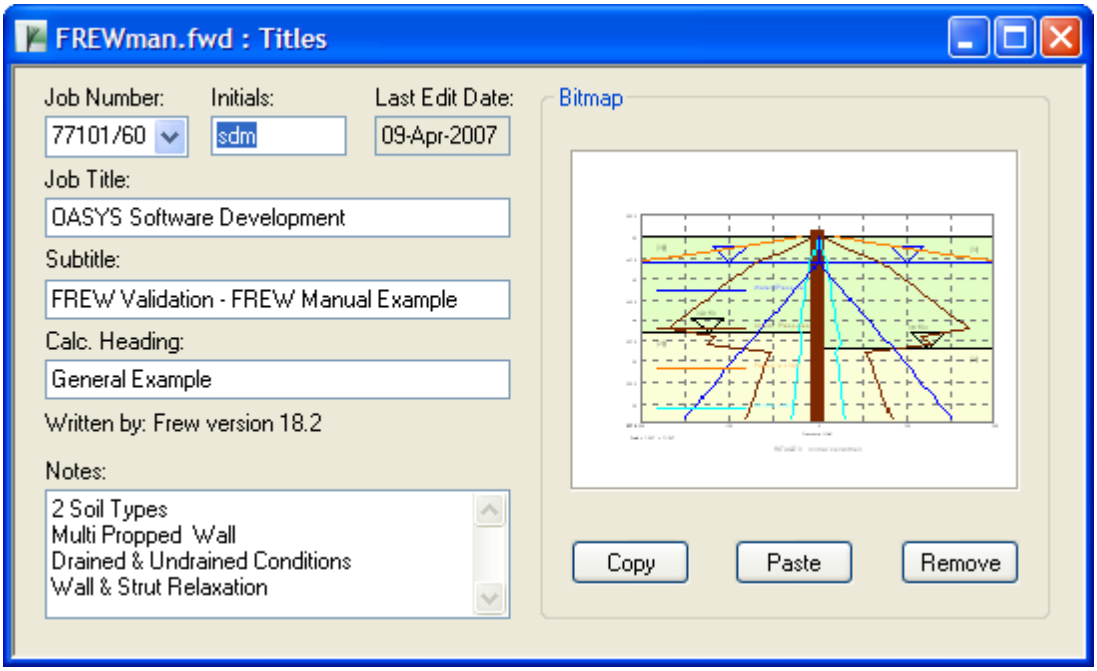
Note: The location of the nodes can **not** be changed in a later stage.

It is useful to sketch out the problem from beginning to end to ensure that the correct parameters are entered as global data, see Assembling Data.

Note: Tables are locked for editing in the program when results are available. To edit the data in the tables, the user has to explicitly delete the results.

3.4.1 Titles

When a existing file is opened, or a new file created without the New Model Wizard, the first window to appear is the **Titles** window.



This window allows entry of identification data for each program file. The following fields are available:

© Oasys Ltd. 2014

Job Number	allows entry of an identifying job number. The user can view previously used job numbers by clicking the drop-down button.
Initials	for entry of the users initials.
Date	this field is set by the program at the date the file is saved.
Job Title	allows a single line for entry of the job title.
Subtitle	allows a single line of additional job or calculation information.
Calculation Heading	allows a single line for the main calculation heading.

The titles are reproduced in the title block at the head of all printed information for the calculations. The fields should therefore be used to provide as many details as possible to identify the individual calculation runs.

An additional field for **notes** has also been included to allow the entry of a detailed description of the calculation. This can be reproduced at the start of the data output by selection of notes using File | Print Selection.

3.4.1.1 Titles window - Bitmaps

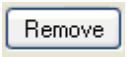
The box to the left of the Titles window can be used to display a picture beside the file titles.

To add a picture place an image on to the clipboard. This must be in a RGB (Red / Green / Blue) Bitmap format.

Select the

button to place the image in the box.

The image is purely for use as a prompt on the screen and can not be copied into the output data. Care should be taken not to copy large bitmaps, which can dramatically increase the size of the file.

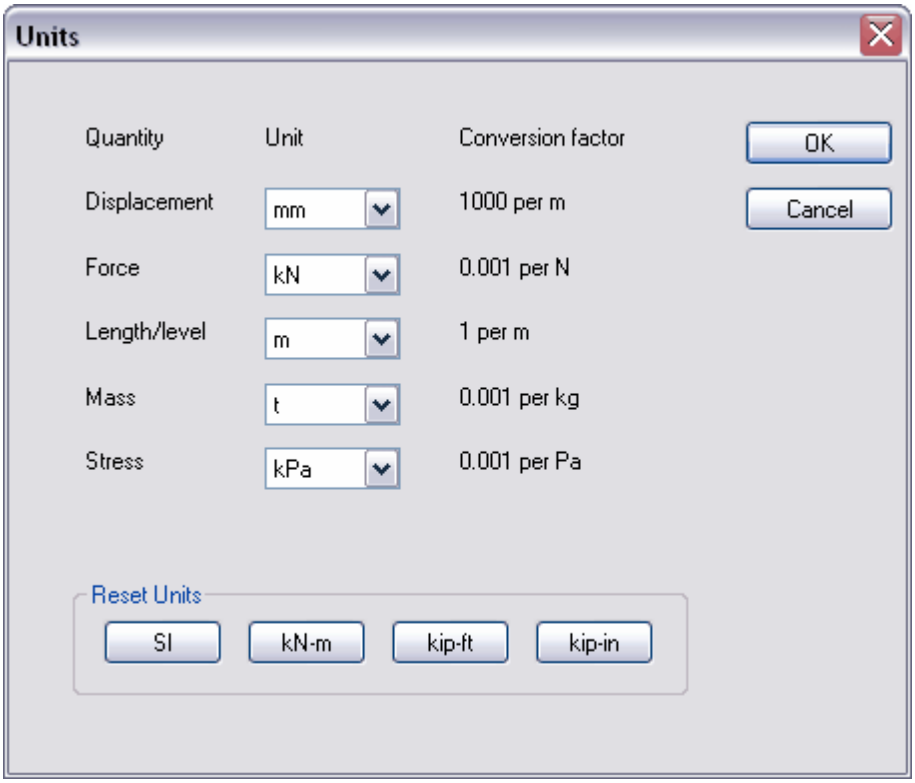
To remove a bitmap select the 

button.

3.4.2 Units

This option allows the user to specify the units for entering the data and reporting the results of the calculations.

© Oasys Ltd. 2014



Default options are the Système Internationale (SI) units - kN and m. The drop down menus provide alternative units with their respective conversion factors to metric.

Standard sets of units may be set by selecting any of the buttons: SI, kN-m, kip-ft or kip-in.

Once the correct units have been selected then click 'OK' to continue.

SI units have been used as the default standard throughout this document.

3.4.3 Material Properties

The properties for the different layers of materials, either side of the wall, are entered in tabular form.

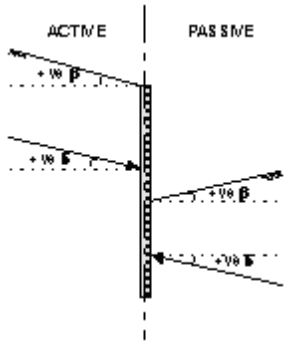
Properties must be entered for all the materials which will be required for all construction stages. If drained and undrained parameters of the same material type are to be used then each set of parameters must be entered on a separate line.

Note: The user should understand the way **Frew** models undrained and drained behaviour and the transition between the two. For further information see the section on Total and Effective Stress.

Brief **descriptions** for each of the material types can be entered here. This description is used when assigning material types to either side of the wall, thereby creating the soil zones (see entering Soil Zones).

Note: Material type 0 represents air or water - no additional input data is required by the user.

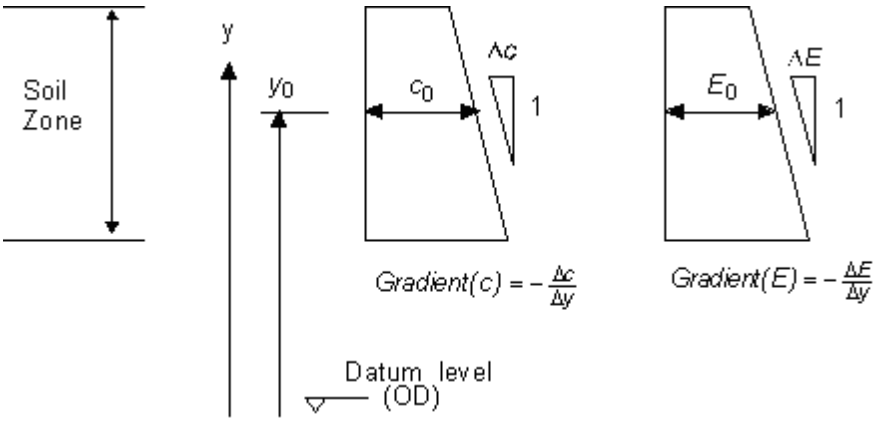
© Oasys Ltd. 2014

Material Property	Description
E0	**Young's modulus** given as 1. A general constant parameter for the layer as a whole, or 2. A specific value for a given reference level y0.
γ	**Unit weight,** defined as the bulk unit weight γ.
K_0	**Coefficient of earth pressure at rest**, i.e. horizontal effective stress / vertical effective stress.
Earth Press. Coef.	Select from the drop-down list whether the earth pressure coefficients will be **"Calculated"** or **"User Specified"**. Calculated User Specified see Calculation of earth pressure coefficients Note: For **"Calculated"** the cells of Ka, Kp, Kac and Kap will be uneditable and when values are entered into ϕ', δ/ϕ', β and Cw/c the earth pressure coefficients will be calculated. For **"User Specified"** the cells for ϕ', δ/ϕ', β Cw/c will be greyed out and the cells of Ka, Kp, Kac and Kap will be editable.
γ	**Unit weight,** defined as the bulk unit weight γ.
ϕ'	Angle of internal friction.
δ/ϕ'	**Ratio** of wall-soil friction angle to shearing resistance angle.
β	**Angle of ground surface** to horizontal in degrees. 
C_w / C	**Ratio** of wall adhesion to soil cohesion.
K_a	**Active earth pressure**, with allowance for soil/wall friction.
K_p	**Passive earth pressure**, with allowance for soil/wall friction.
K_{ac}	**Active earth pressure due to cohesion**. $2 * \sqrt{K_a\left(1+\frac{c_w}{c}\right)}$ or $2 * \sqrt{K_a}$ if $C_w = 0$

© Oasys Ltd. 2014

K_{pc}	**Passive earth pressure due to cohesion**. $2*\sqrt{K_p\left(1+\frac{c_w}{c}\right)}$ or $2 * \sqrt{K_p}$ if $C_w = 0$
K_r	**Ratio of change in horizontal effective stress to a unit change in vertical effective stress**. i.e. ν / (1 - ν) where ν = Poisson's ratio, see Effects of excavation and backfill.
C*0*	**Cohesion** referenced at y0 and taken as either: C' for drained soil or C*u* for an undrained soil.
y0	**Reference level** for the gradient of cohesion (c) or Young's modulus (E) with depth. Tab across the column if they are constant with depth. Note: This level does not have to correspond to the top of the material layer. It is a reduced level and is not referenced from the bottom of the layer.
c gradient	The **rate of change of cohesion** with depth. A positive value means cohesion is increasing with depth.
E gradient	The **rate of change of Young's modulus** with depth. A positive value means stiffness is increasing with depth.
Drained / Undrained	Indicates whether the material is **Drained** or **Undrained**. Note: This setting is only used by Partial factors and Passive softening to access drained/undrained soil strength properties.
Shape factor	For undrained materials only: factor to use in weighting the failure envelope on the dry side between Mohr-Coulomb and Modified Cam-Clay envelopes. Default is 1. Used only in calculation of undrained pore pressures, see Undrained Materials and Calculated Pore Pessures.
Material no. for effective stress parameters	For undrained materials only: the number of the material from which to use effective stress parameters in undrained pore pressure calculations. **Note:** the user should set the last column to zero if undrained pore pressure calculations are not required.

© Oasys Ltd. 2014



$c = c_0 + Grad(c)*(y_0 - y)$

$E = E_0 + Grad(E)*(y_0 - y)$

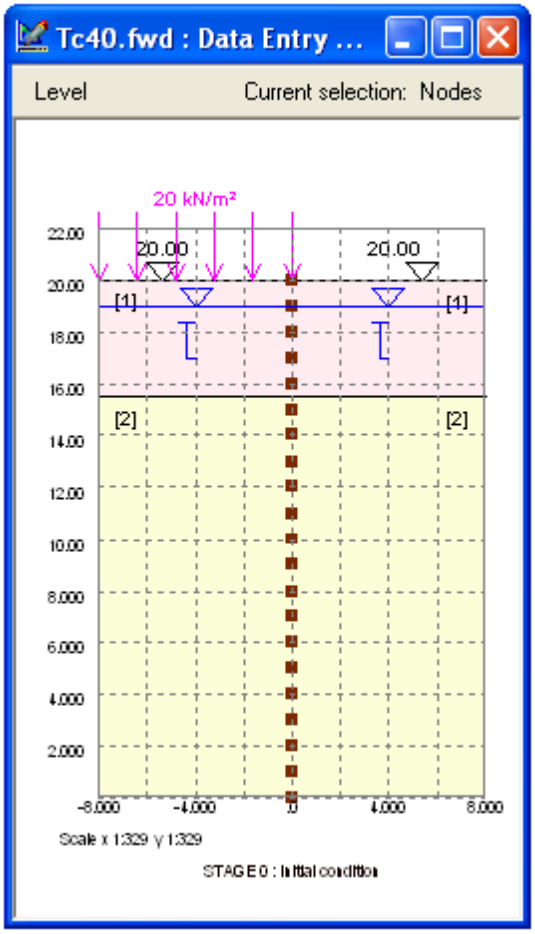
3.4.4 Nodes

The Node level entry data is only available if automatic node generation is switched off in the New Model Wizard or the Node generation data dialog. Nodes can then be entered by using the graphical or tabular display and are required at the following locations:

1. Strut levels.
2. Top and base of the wall and levels at which the wall stiffness (EI value) changes.
3. Levels either side of the ground surfaces during excavation back fill and the interfaces between soil zones.

Note 1: Ground surfaces and soil zone interfaces occur midway between nodes. The exception is the highest ground surface which can coincide with the top node.
Note 2: Where seismic analysis is undertaken, and the seismic force is applied as a point load, struts will be generated at the location in which the forces are to be applied. As a result nodes are also required at these levels.

© Oasys Ltd. 2014



© Oasys Ltd. 2014

Tc40.fwd : Node Data

	A	B	C
Node	Level [m]	Left material	Right material
Defaults		None	None
1	20.00	Material 1	Material 1
2	19.00	Material 1	Material 1
3	18.00	Material 1	Material 1
4	17.00	Material 1	Material 1
5	16.00	Material 1	Material 1
6	15.00	Material 2	Material 2
7	14.00	Material 2	Material 2
8	13.00	Material 2	Material 2
9	12.00	Material 2	Material 2
10	11.00	Material 2	Material 2
11	10.00	Material 2	Material 2
12	9.00	Material 2	Material 2
13	8.00	Material 2	Material 2
14	7.00	Material 2	Material 2
15	6.00	Material 2	Material 2
16	5.00	Material 2	Material 2
17	4.00	Material 2	Material 2
18	3.00	Material 2	Material 2
19	2.00	Material 2	Material 2
20	1.00	Material 2	Material 2
21	0.00	Material 2	Material 2
22			

In addition to placing nodes at key locations in the construction sequences, it is also important to space them at reasonably regular, close intervals down the line of the wall and beyond to the base of the problem. This allows the program to clearly model the flexibility of the wall and provide results of the forces, pressures and bending moments which are given at each node location.

Note: As a guide it is recommended that the maximum separation between any two nodes must never be greater than twice the minimum separation between any of the nodes. **Frew** gives a warning if this rule is violated.

Frew may have difficulty if unusually short, stiff elements are used. To obtain equilibrium, bending moments are obtained from the calculated curvature (second differential of displacement). This requires that for any three consecutive nodes, curvature can be derived to sufficient accuracy from displacements which are computed to six significant figures. This becomes difficult for short stiff elements especially if they displace by large amounts (e.g. at the top of a flexible wall).

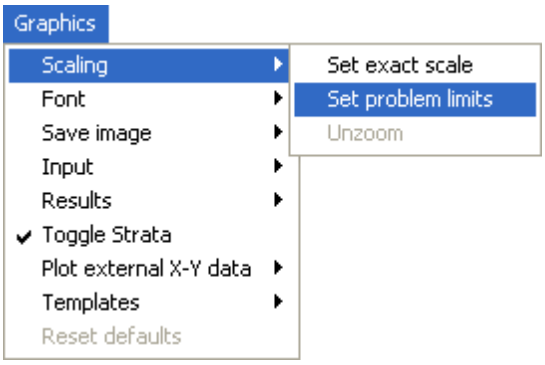
Note: Material properties must be defined before the program allows the node locations to be selected.

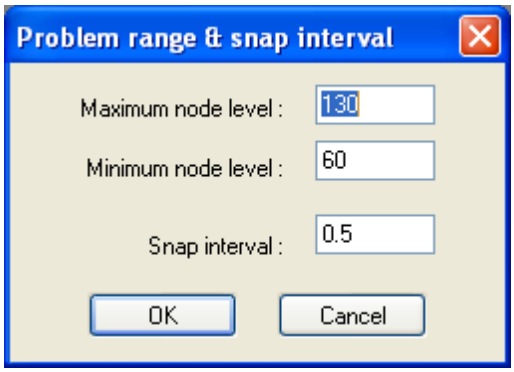
Defining the depth of the problem

If the correct level range is not shown on the graphical view, define the extent of the problem by selecting the menu option Graphics | Scaling | Set Problem limits and then enter the maximum and minimum levels of the nodes.

© Oasys Ltd. 2014

Problem Range & Snap Interval





Note: When soil stiffness is represented using the "**Safe**" or Mindlin methods, it is assumed that the lowest node specified in the data defines a **horizontal rough rigid boundary**.

The snap interval is also entered here. This provides the closest point onto which the cursor will lock to mark a point. The snap interval is taken as the nearest interval in metres.

Adding nodes.

To add nodes select the Global data | Node levels menu option or select the nodes button on the graphics toolbar. Nodes can then be added by entering their level directly in the table or graphically by the following procedure;

1. Use the mouse to place the cursor over the location of the top node.
2. Click with **left button** on the location.

This will place the node on the line of the wall.

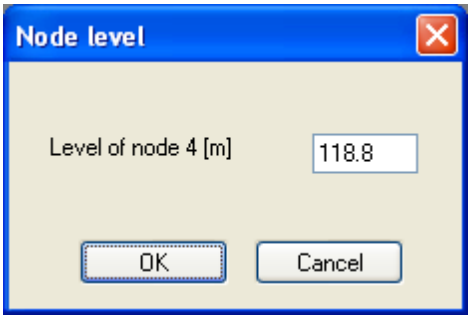
Note: If the scale of the diagram is too small to locate the nodes accurately, then maximise the main and graphics windows to increase the size of the image.

Editing the location of the node.

If the location of the node requires editing then this can be done in one of two ways:

1. Place the cursor over the node and click the right mouse button. This will bring up an amendment box.

© Oasys Ltd. 2014



Amend the level of the node and select OK.

2. As an alternative go to the table at the side of the diagram. Use the arrow keys and return button or the mouse to select the appropriate node. Change the value and press enter.

Deleting nodes.

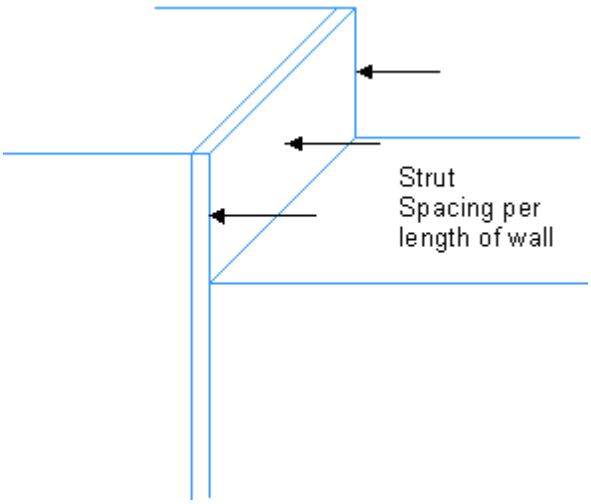
Nodes can be deleted by;

1. Placing the cursor over the correct node and then using the left mouse button whilst simultaneously holding down the Shift key.
2. Highlighting the line number in the table using the left mouse button, then pressing the Delete key.

3.4.5 Strut Properties

Struts may be inserted (or removed) at any node at any stage in the analysis. All the struts required for the various stages of analysis, however, must be inserted in the global data.

Struts and **anchors** are modelled in terms of an "equivalent strut" which represents the total number of struts present in a one m length of wall (e.g. for struts at 2m centres, input half the force and stiffness of an individual one).



For each equivalent strut, the following items are required:

© Oasys Ltd. 2014

FREWman.fwd : Struts

Strut	A	B	C	D	E	F	G
	Stage		Node number	Prestress [kN/m]	Stiffness [kN/m/m]	Angle [°]	Lever arm [m]
	In	Out					
Defaults							
1	3	6	2	200.00	0.00	40.00	0.50
2	4	6	2	0.00	2000.00	40.00	0.50
3	6	11	1	0.00	40000.00	0.00	0.00
4	6	11	10	0.00	40000.00	0.00	0.00
5	11		1	0.00	20000.00	0.00	0.00
6	11		10	0.00	20000.00	0.00	0.00
7							

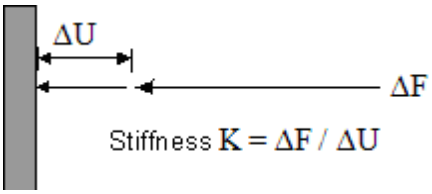
- The stage number at which the strut is inserted
- The stage number at which the strut is removed (leave blank if the strut is still active during the final stage)
- The level at which the strut acts (if using automatic node generation) or the node number (if using manual node entry).
- Any pre-stress force applied i.e. force in the strut when it is inserted.
- Its Its stiffness i.e. change in force (ΔF kN) in strut for a unit (ΔU = 1m) movement at it's point of application, both measured along the axis of the strut, per m run of wall.

  $K = \Delta F/\Delta U = AE/L$

  A = Cross sectional area of strut

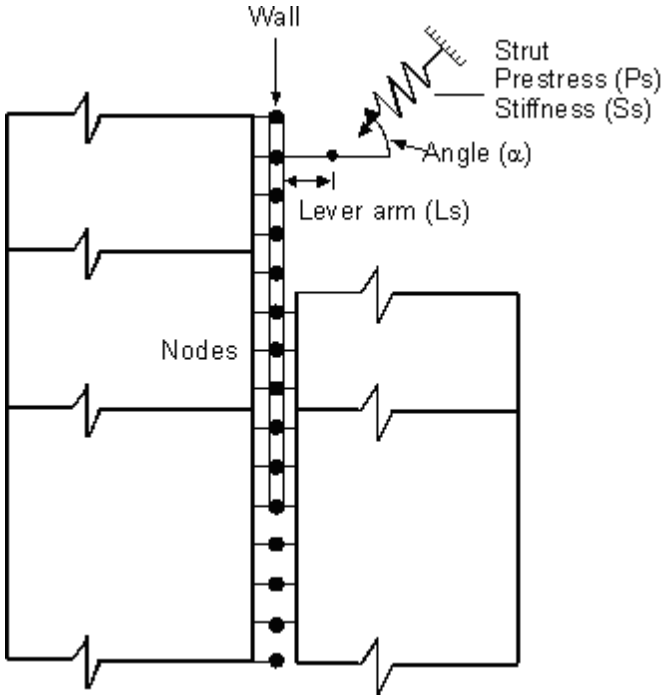
  E = Young's modulus

  L = Length of strut



- The angle (degrees) to the horizontal (+ve anticlockwise).
- A moment can be applied to the wall by using a strut at an angle of 90 degrees and a lever arm, i.e. distance from neutral axis of wall to point of application of strut (+ve to the right).

© Oasys Ltd. 2014



Note: More than one strut may be defined at a particular node, and not all struts need to act simultaneously.

If a strut is inserted in **Stage 0**, prior to the wall being installed, only the horizontal pre-stress force is modelled. The stiffness of the strut will be modelled in subsequent stages.

An **applied moment** or a **moment restraint** at a node can be modelled by using a strut with an inclination of 90 degrees and a non-zero lever arm together with an applied pre-stress force or a stiffness respectively.

3.4.5.1 Modelling of Anchors

Stressing an anchor may be modelled by specifying a strut with a pre-stress force equal to the stressing force and a zero stiffness. The stiffness of zero would maintain a constant force at the point of application throughout the analysis.

In subsequent stages after the anchor is 'locked off' it is usually convenient to remove this strut and insert a strut that models both the pre-stress and stiffness of the anchor.

Inclined anchors are modelled by specifying an inclination to the horizontal and if they are not applied at the vertical axis of the wall a lever arm can be specified to allow for this, see Strut Properties.

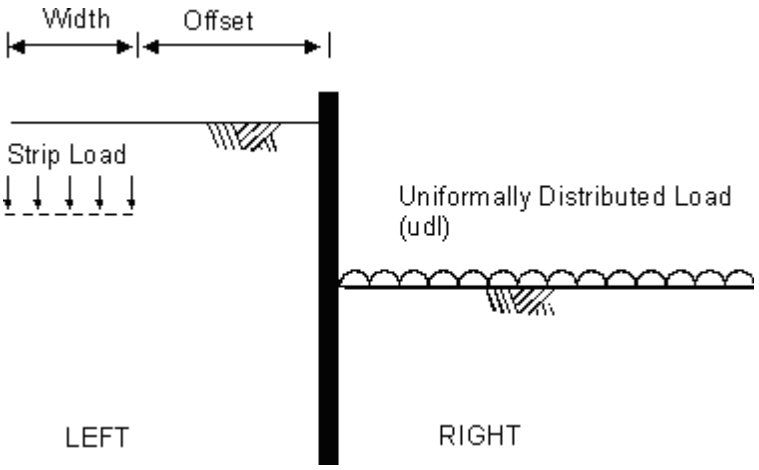
© Oasys Ltd. 2014

3.4.6 Surcharges

Surcharges may be applied at or below the surface of the ground on either side of the wall.

These are always uniform pressures and may be in the form of;

1. Strip loads of any width running parallel to the wall or
2. Uniformly distributed loads (udl) of infinite extent.



The input data required for each surcharge is as follows.

FREWman_V19.1.fwd : Surcharges

	A	B	C	D	E	F	G	H	I	J
Surcharge	Stage		Side	Level [m]	Type	Pressure [kN/m²]	Partial Factor	Offset [m]	Width [m]	Ks
	In	Out								
Defaults			Left		UDL		1.00			
1	2	10	Left	50.00	UDL	10.00	1.10			
2	6		Right	40.00	UDL	15.00	1.00			
3										

- The stage number at which the surcharge is applied
- The stage number at which the surcharge is removed (leave blank if the surcharge is still present in the final stage)
- The side (left or right) of the wall where the surcharge is to be located.
- The level.
- The load type (UDL or strip load)
- The pressure.
- The partial factor to be applied to the surcharge. This will be applied only is partial factor analysis is specified by the user in "Partial Factors" dialog.
- Dimensions:
  Offset: Distance of the line of the wall to the nearest edge of the strip plus
  Width of the strip normal to the line of the wall.
  These fields are not available for a udl.
- Ks, used as discussed below:
  = 1 if the surcharge is narrow compared to the underlying soil layer
  = Kr if the surcharge is very wide, see Material Properties.

Note: The chosen value of Kr to which Ks is equated, should correspond to the material type that is most influenced by the transfer of the applied load to the wall. If the layers are relatively thin then an average value should be taken. If the stages include a change between undrained and drained

© Oasys Ltd. 2014

materials then multiple surcharges should be entered to take the change of Kr (at the relevant stage) into account.

The surcharge can be applied before the wall is inserted (Stage 0). If this is the case the program computes the effects of the surcharge on the soil stresses before installation of the wall. In some cases this may however prevent the program from converging as there is a discontinuity in the lateral earth pressures at the top node. Where this occurs this can usually be worked around by applying the surcharge at a level slightly below the ground surface.

3.4.6.1 Application of Uniformly Distributed Loads

To determine the elastic effect on the horizontal effective stress, udl surcharges are multiplied by K_0 in Stage 0, whereas in later stages they are multiplied by K_r.

They are therefore treated in the same way as the weight of soil, both in the initial state (Stage 0) and in later stages as excavation and filling takes place, see Effects of excavation and backfill. In all stages, including Stage 0, strip surcharges are multiplied by the factor Ks described above.

Active and passive limit pressures are also modified using the values of K_a and K_p for each layer beneath the udl.

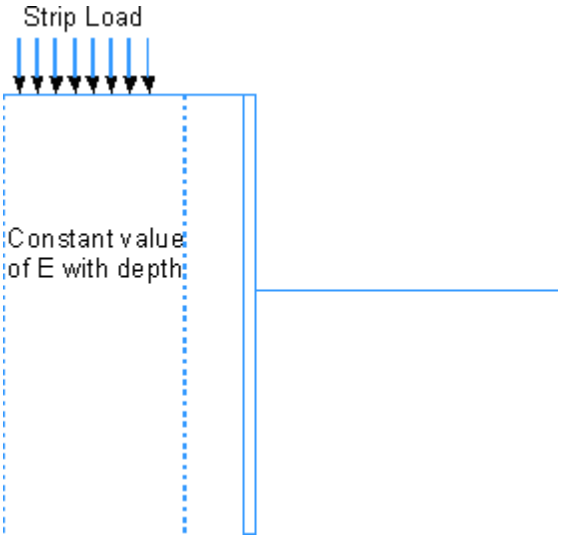
3.4.6.2 Application of Strip Loads

For strip loads the change in stress in elastic conditions is difficult to determine because the horizontal stress is extremely sensitive to the variation with depth of the soil stiffness and to anisotropy of the soil stiffness. Two extremes have therefore been considered.

1. For the case where the Young's modulus (E) for the soil is constant for a depth several times greater than the width of the surcharge the Boussinesq equations may be used to derive horizontal stresses in the ground.

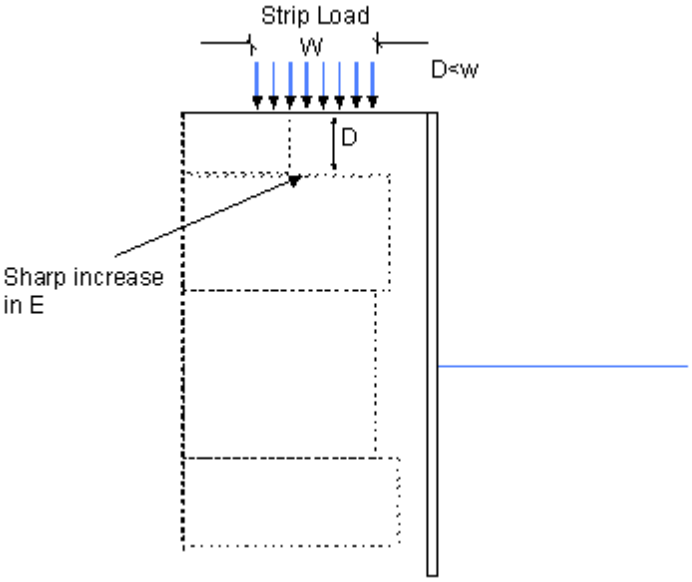
   The pressures therefore on a rigid (ideally frictionless) vertical boundary would be double the Boussinesq values.

© Oasys Ltd. 2014



2. For the case where the stiffness (E) increases sharply at a depth less than the width of the surcharge, the load will appear to the more flexible soil to act rather like a 'udl'.

   For the stiffer soil the effect of the surcharge load will still appear as the Boussinesq pressure.



For both cases the analysis calculates the change of pressure on the wall before further movement using the equation

$$p = 2K_s\sigma_{phB}$$

where:

σ_{phB} = the change of horizontal stress according to the Boussinesq equations

K_s = is a correction factor specified by the user. Where Young's modulus is constant with

© Oasys Ltd. 2014

depth, Ks should be taken as 1.0.

For the case where stiffness increases sharply K_s can have a large range of values, the evaluation of which is beyond the scope of this text. However if the strip load is wide compared with its distance from the wall and the depth of the deforming soil, a value of $K_s = \nu/(1 - \nu)$ will give results equivalent to loading with a udl with $K_r = \nu/(1 - \nu)$.

Active and Passive Pressures
The values of active and passive pressures due to strip loads parallel to the wall are discussed in Active Pressures due to Strip Load Surcharges and Passive Pressures due to Strip Load Surcharges.

The method described for the active pressure is automatically applied by the program, **but the method described for the passive pressure is not applied.** The user must therefore manually enhance the passive pressure coefficient K_p or the soil cohesion c, if this effect is to be incorporated into the analysis, see Passive Pressures due to Strip Load Surcharges.

The user is recommended to study the graphical output and check whether the pressures adopted by the program are acceptable.

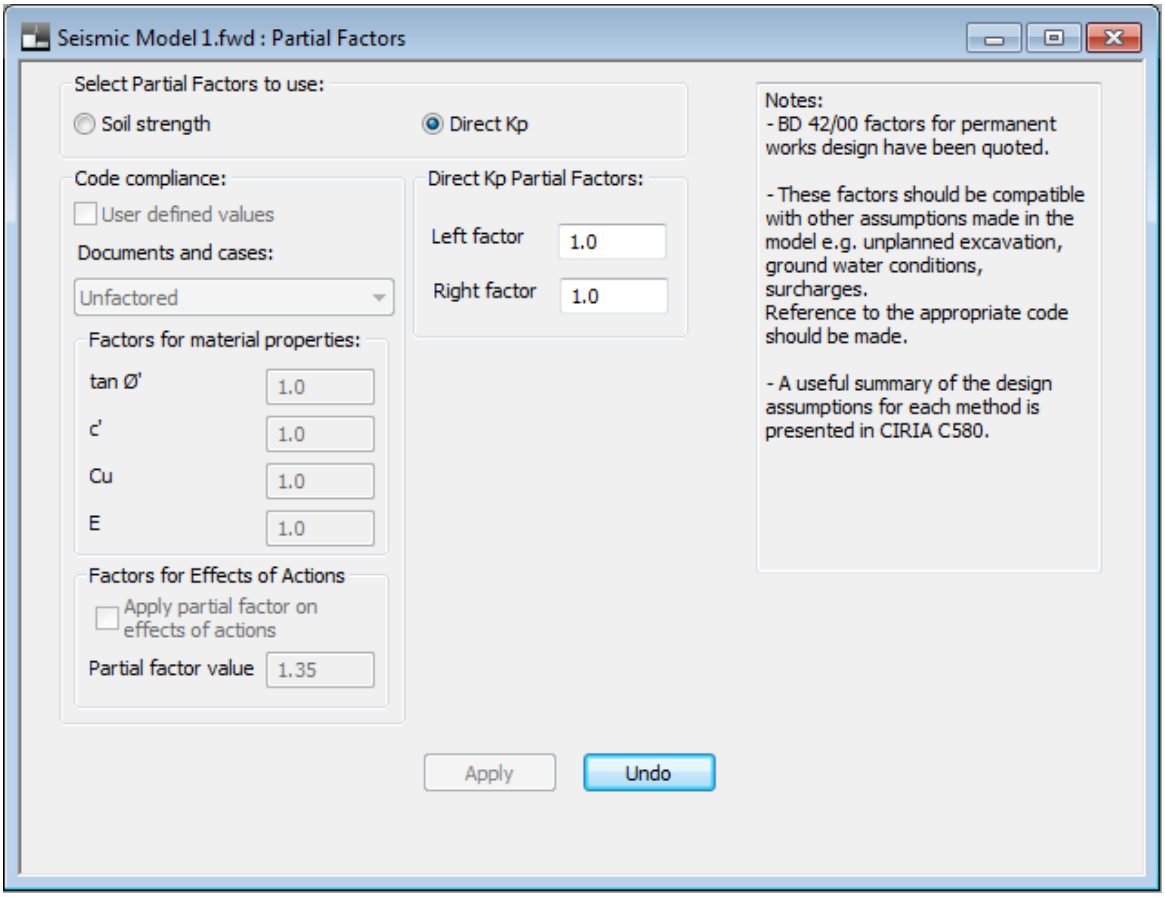
3.4.7 Partial Factors

The purpose of factors is to allow for uncertainty in material properties, loading and calculation models and to ensure safety and acceptable performance.

These are global factors that are applied to material properties or surcharges input parameters. The new material parameters affected by these factors will then be used in the calculations. A single set of factors shall be selected and these will apply to all materials in all stages.

WARNING: **Frew** has had new features added to simplify application of partial factors in line with EC7. However, there are alternative ways of complying with EC7 including manual adjustment of certain values. The features in the program do not automatically make a design EC7 compliant and the user must continue to check the output carefully to ensure the assumptions and adjustments to characteristic values are as they require. Note that pore pressures and strut pre-stress are not factored. If a strut pre-stress is used to model a structural force, and other effects of actions are being factored, the user may wish to factor the input value of strut pre-stress.

© Oasys Ltd. 2014

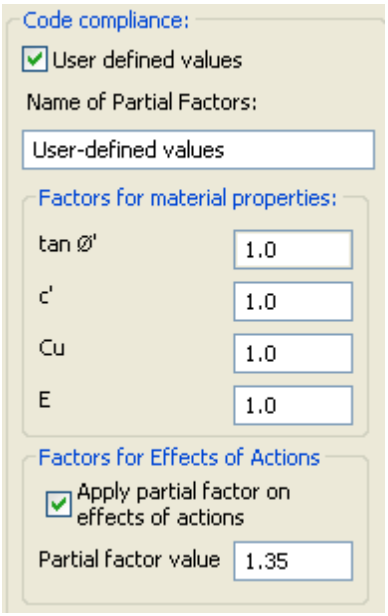


Soil strength Partial Factors can only be applied if automatic calculation of earth pressure coefficients is selected for all materials. see: Material properties

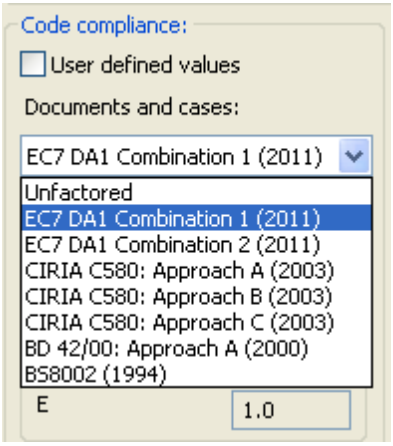
Note: When soil strength factors are selected the direct Kp Partial Factors can not be selected and are set to 1.0.

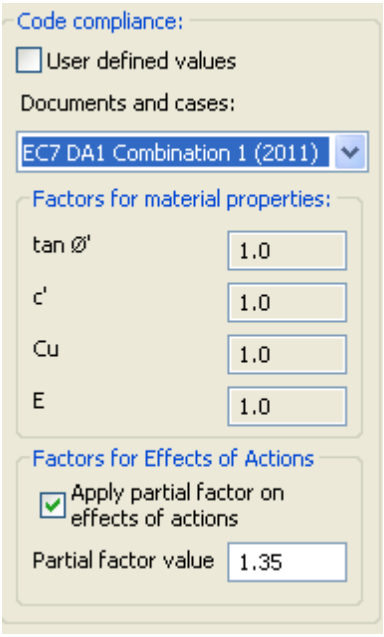
User defined values are a set of soil strength Partial Factors that can be named and set to any value by the user.

© Oasys Ltd. 2014



Document and cases is a pre defined list of code compliance Partial Factors:

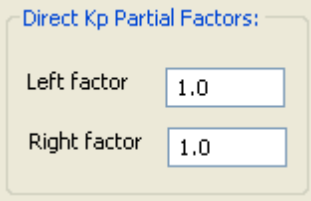




Direct Kp Partial Factors can be applied even if earth pressure coefficients are User Specified.

Note: When this set of Partial Factors is selected the Soil strength factors can not be selected and are set to "Unfactored" and to 1.0.

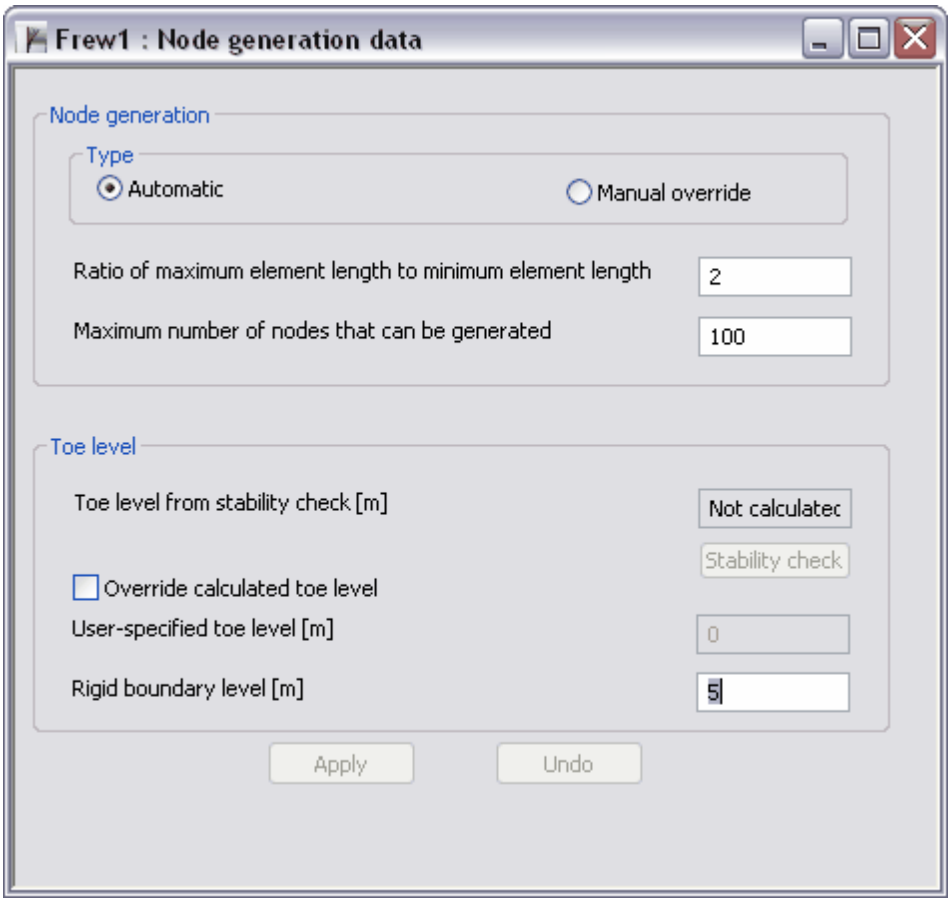
© Oasys Ltd. 2014



Surcharges Partial Factors are applied to each type of load (UDL or Strip) and directly in the Surcharges table.

3.4.8 Node generation data

This dialog is available from the Global data menu or the Gateway.



It allows setting of the node generation method (automatic or manual), some settings for automatic node generation, and whether to calculate the wall toe level from a stability check. If a stability check has already been carried out, this dialog will show the calculated toe level. This can be overriden by the user.

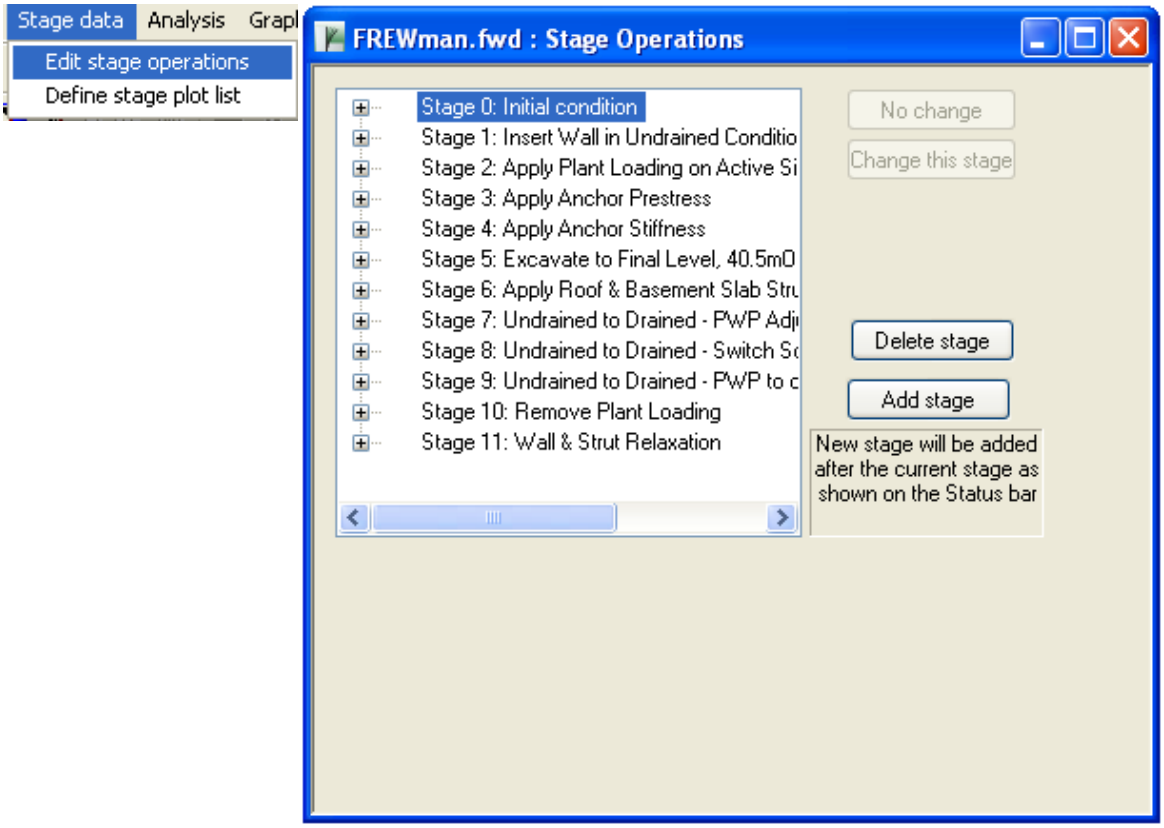
© Oasys Ltd. 2014

3.5 Stage Data

The **Stage Data** menu allows the data to be modified for individual stages using the **Stage Operations** window. This opens a tree diagram, as shown, which then allows access to all available options for each stage. Ticks are placed against those options which have been changed.

This window also allows the creation of new stages and the deletion of those no longer required. When "Add stage" is selected the new stage can be inserted after any existing stage.

Parameters can also be set to change in a particular stage or not to change.


Stage data
Analysis
Grapl
Edit stage operations
Define stage plot list
FREWman.fwd : Stage Operations
Stage 0: Initial condition
Stage 1: Insert Wall in Undrained Conditio
Stage 2: Apply Plant Loading on Active Si
Stage 3: Apply Anchor Prestress
Stage 4: Apply Anchor Stiffness
Stage 5: Excavate to Final Level, 40.5mO
Stage 6: Apply Roof & Basement Slab Stru
Stage 7: Undrained to Drained - PWP Adj
Stage 8: Undrained to Drained - Switch Sc
Stage 9: Undrained to Drained - PWP to c
Stage 10: Remove Plant Loading
Stage 11: Wall & Strut Relaxation
No change
Change this stage
Delete stage
Add stage
New stage will be added after the current stage as shown on the Status bar

Note: Left click on the boxes and to open or close the tree diagram for each stage.

As mentioned earlier, the user can access stage specific data of the **current** stage using the Gateway.

Note:

© Oasys Ltd. 2014

3.5.1 Stage 0 - Initial Conditions

The information must first be set for Stage 0 - the initial conditions before entry of the wall in Stage 1. Stage 0 appears automatically in the summary tree diagram on creating a new file.

The individual data for each stage can be accessed by using the mouse double left click on the data heading in the tree diagram.
This action opens the window for data input.

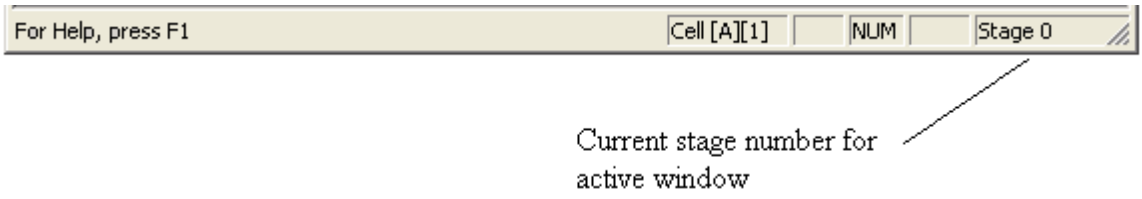
The following data must be entered to allow the calculation of stage 0

	Global Data	**Stage 0 Data**
Compulsory	Material properties	Soil zones (drained materials)
	Node levels	
	Surcharges	Analysis Method
	Struts	
		Convergence control parameters
Optional		Water data

Note: The properties set in Stage 0 will be carried forward into subsequent stages unless otherwise amended.

On completion of Stage 0 new stages can be added or inserted to the list.

The number of the current stage is always displayed in the status line at the base of the main window.



3.5.2 New Stages

New stages can be added to the list by selecting Add Stage on the Stage Operations window. This activates a 'New Stage Title' box.

© Oasys Ltd. 2014

The stage title is then entered and the number of the stage before the new stage. Once the OK button is selected the new stage is added.

Note : The number that first appears in the Inserting after Stage window is the number of the stage currently highlighted by the cursor.

3.5.3 Inserting Stages

Select the "Add stage" button on the stage operations tree diagram and follow the instructions as for new stages.

3.5.4 Deleting a Stage

Stages can be deleted by highlighting the stage title in the Stage Operation window and selecting Delete stage. A check box will appear before the stage is deleted.

3.5.5 Editing Stage Data

It is possible to step through the stages in order to access and edit the same data window for each stage. Use the buttons on the tool bar to move up and down between the various stages.

The number of each stage is displayed on the status line at the base main window.

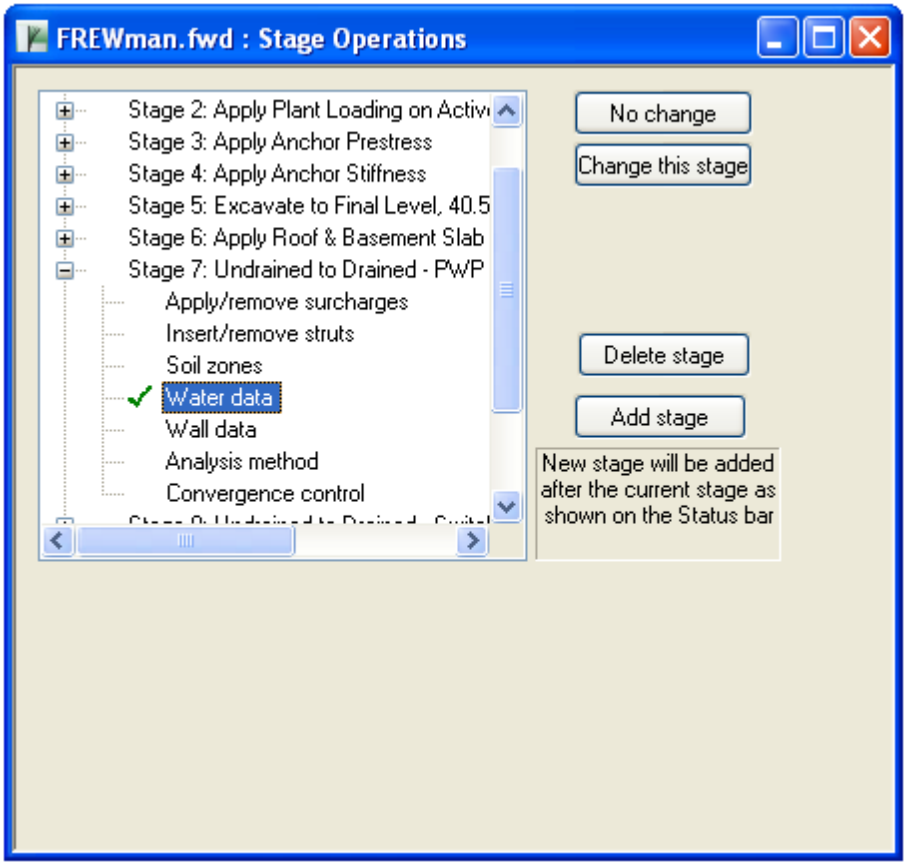
Once the correct stage has been reached. edit the data as normal.

To reach specific windows go to the Stage Operations tree diagram, highlight the required operation at the required stage and then either

- double click on the highlighted operation or
- select the "Change this stage" button

to open the window.

© Oasys Ltd. 2014

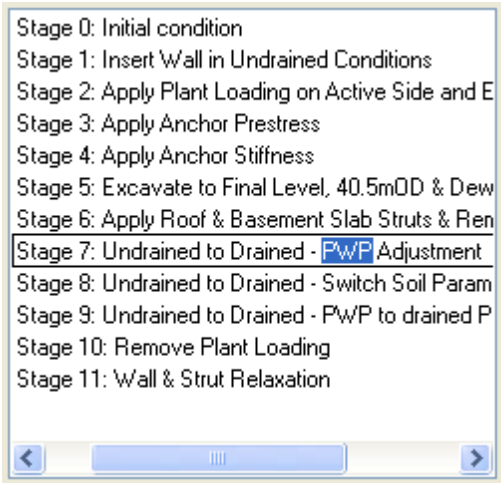

FREWman.fwd : Stage Operations
Stage 2: Apply Plant Loading on Activ
Stage 3: Apply Anchor Prestress
Stage 4: Apply Anchor Stiffness
Stage 5: Excavate to Final Level, 40.5
Stage 6: Apply Roof & Basement Slab
Stage 7: Undrained to Drained - PWP
Apply/remove surcharges
Insert/remove struts
Soil zones
Water data
Wall data
Analysis method
Convergence control
No change
Change this stage
Delete stage
Add stage
New stage will be added after the current stage as shown on the Status bar

Note: Changes made in a stage will be copied through to subsequent stages until the program encounters a specific change already made by the user. For example, changing the soil zones in an early stage will update later stages which had the same soil zone specification.

3.5.6 Editing Stage Titles

The stage titles can be edited by left clicking on the title so that it becomes highlighted in yellow and then clicking again to get the cursor before typing the amendments as required.

© Oasys Ltd. 2014


Stage 0: Initial condition
Stage 1: Insert Wall in Undrained Conditions
Stage 2: Apply Plant Loading on Active Side and E
Stage 3: Apply Anchor Prestress
Stage 4: Apply Anchor Stiffness
Stage 5: Excavate to Final Level, 40.5mOD & Dew
Stage 6: Apply Roof & Basement Slab Struts & Ren
Stage 7: Undrained to Drained - PWP Adjustment
Stage 8: Undrained to Drained - Switch Soil Param
Stage 9: Undrained to Drained - PWP to drained P
Stage 10: Remove Plant Loading
Stage 11: Wall & Strut Relaxation

3.5.7 Apply/Remove Surcharges

Surcharges can be applied and removed individually for each stage. Edit the Stage In/Out entries as required in the table.

FREWman_V19.1.fwd : Surcharges

	A	B	C	D	E	F	G	H	I	J
Surcharge	Stage		Side	Level [m]	Type	Pressure [kN/m²]	Partial Factor	Offset [m]	Width [m]	Ks
	In	Out								
Defaults			Left		UDL		1.00			
1	2	10	Left	50.00	UDL	10.00	1.10			
2	6		Right	40.00	UDL	15.00	1.00			
3	7	8	Left	35.00	Strip	18.00	1.20	5.00	3.00	0.30
4										

3.5.8 Insert/Remove Struts

Struts can be inserted and removed individually for each stage. Edit the Stage In/Out entries as required in the table.

FREWman.fwd : Struts

	A	B	C	D	E	F	G
Strut	Stage		Node number	Prestress [kN/m]	Stiffness [kN/m/m]	Angle [°]	Lever arm [m]
	In	Out					
Defaults							
1	3	6	2	200.00	0.00	40.00	0.50
2	4	6	2	0.00	2000.00	40.00	0.50
3	6	11	1	0.00	40000.00	0.00	0.00
4	6	11	10	0.00	40000.00	0.00	0.00
5	11		1	0.00	20000.00	0.00	0.00
6	11		10	0.00	20000.00	0.00	0.00
7							

© Oasys Ltd. 2014

3.5.9 Soil Zones

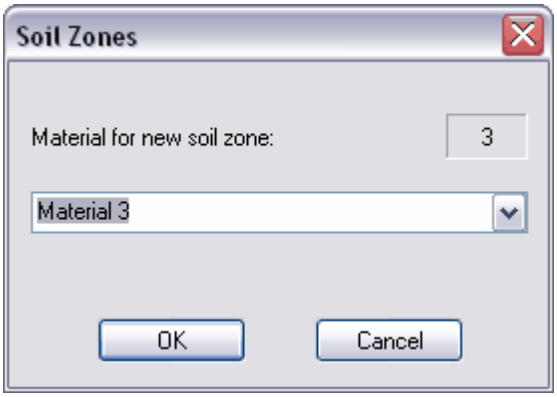
The layers of soil on either side of the wall are described in terms of Soil Zones. They are added by tabular or graphical input and are shown graphically in the general display of the layout of the wall.

Note: The interfaces between the soil zones are set **midway between nodes**. If automatic node generation is being used, the program will do this for you. Otherwise, the locations of soil zone interfaces must be taken into account when defining the node positions.

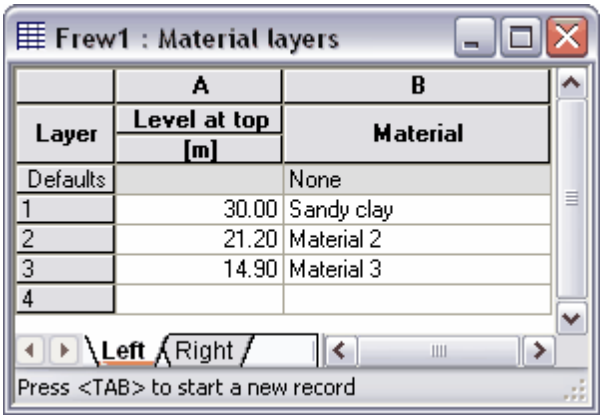
To add soil zones select the soil zones button on the graphics toolbar or the soil zones option from the relevant stage in the 'Stage Operations' window.

Adding/editing soil zones when using automatic node generation

To add a soil zone: Left-click on the graphical input view at the required level of the soil zone interface and choose the required material from the dropdown list in the dialog which appears.



Click OK and the graphical input will be redrawn with the new soil zone shown. Alternatively, the data can be added to the Material Layers table which will open at the same time as the Graphical Input view for this option.

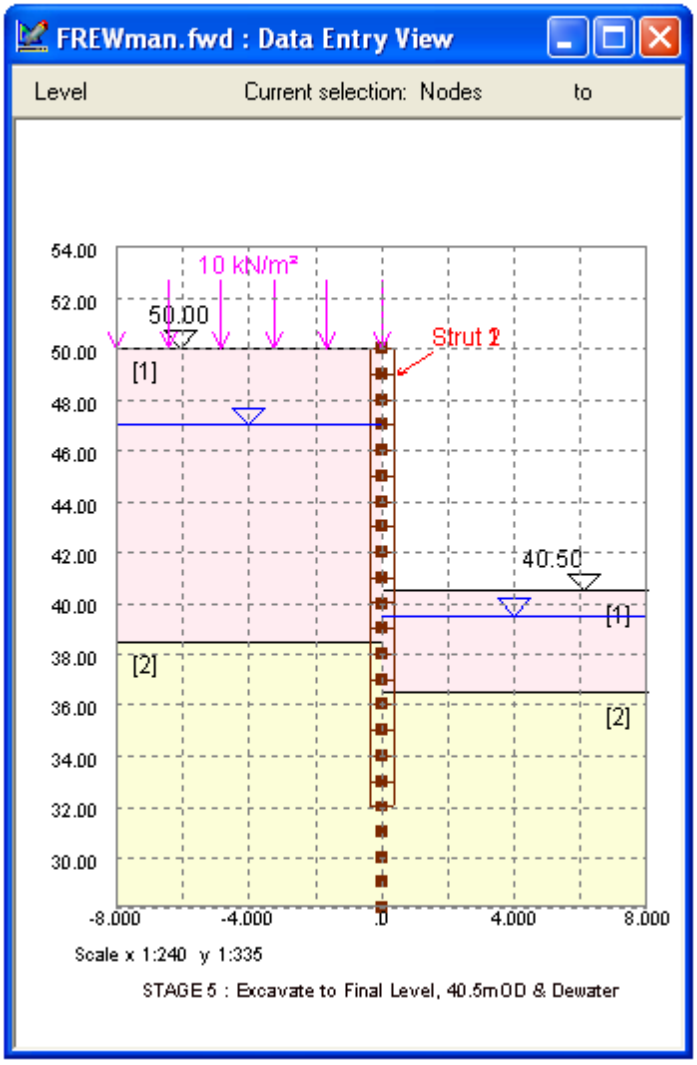


To edit a soil zone's material, right-click in the zone and choose the new material from the dialog. To change the level of a soil zone, or delete it, simply edit or delete the record in the Material Layers table. Note that there is a toolbar shortcut to specify excavation or filling.

Adding soil zones when using manual node entry

.

© Oasys Ltd. 2014

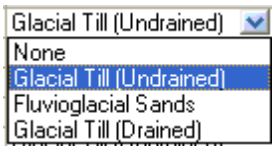


© Oasys Ltd. 2014

FREWman.fwd : Node Data

Node	A Level [m]	B EI below node [kNm2/m]	C Left material	D Right material
Defaults		0.00	None	None
1	50.00	500000.10	Glacial Till (Undrained)	None
2	49.00	500000.10	Glacial Till (Undrained)	None
3	48.00	500000.10	Glacial Till (Undrained)	None
4	47.00	500000.10	Glacial Till (Undrained)	None
5	46.00	500000.10	Glacial Till (Undrained)	None
6	45.00	500000.10	Glacial Till (Undrained)	None
7	44.00	500000.10	Glacial Till (Undrained)	None
8	43.00	500000.10	Glacial Till (Undrained)	None
9	42.00	500000.10	Glacial Till (Undrained)	None
10	41.00	500000.10	Glacial Till (Undrained)	None
11	40.00	500000.10	Glacial Till (Undrained)	Glacial Till (Undrained)
12	39.00	500000.10	Glacial Till (Undrained)	Glacial Till (Undrained)
13	38.00	500000.10	Fluvioglacial Sands	Glacial Till (Undrained)
14	37.00	500000.10	Fluvioglacial Sands	Glacial Till (Undrained)
15	36.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
16	35.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
17	34.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
18	33.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
19	32.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
20	31.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
21	30.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
22	29.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
23	28.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
24				

Soil zone data can be specified using the table by highlighting a cell and selecting the material type from the list presented in the drop down box..



Alternatively, soil zone data can be entered graphically using the procedure described below.

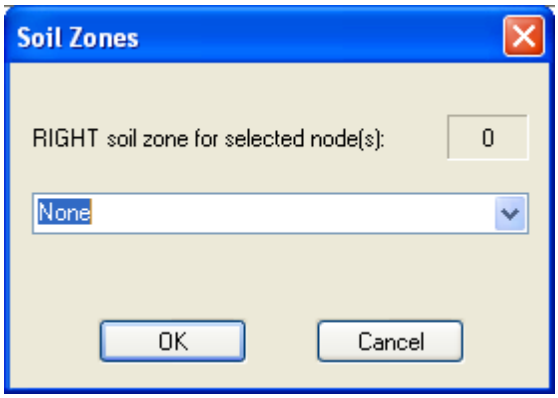
Select the **required range of nodes** on the wall:

1. Place the cursor over the first node and select with the left button.
2. Hold down the Shift key.
3. Place the cursor over the last node (still holding down the Shift key) and select the area of nodes with the left button.

© Oasys Ltd. 2014

To **designate the soil zones**

1. Place the cursor to the left or right of the wall as required.
2. Click the right button on the mouse. This will activate the zones box



Select the required soil from the combo box and select OK.

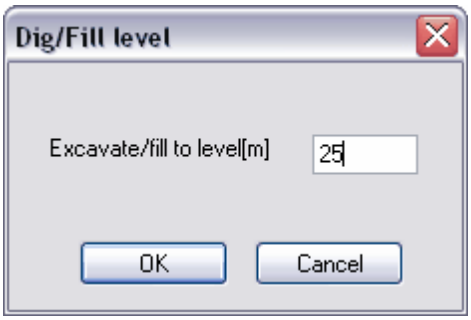
Note: The number of the soil comes from the list of material types created in the materials table, see Material Properties. **Air or water** are designated as material type 0.

Editing soil zones

Once entered the soil zones can be edited using either of the methods given above.

3.5.9.1 Dig/Fill Operations

When using **automatic node generation**, to specify excavation or backfill, click the button on the toolbar. Right-clicking on the left or right side of the wall in the graphical input view will bring up the Dig/Fill dialog.



Enter the required new ground level and click OK. If the new ground level is above the existing ground level (i.e. filling), the uppermost material will be extended to the new ground level.

When using **manual node generation**, excavation is specified by changing the material to the required side of one or more nodes to 0. This represents air or water. Backfilling is specified by

© Oasys Ltd. 2014

changing the material from 0 to the required material number for the backfill material. Digging and backfilling can be carried out on either side of the wall.

Any dig/fill operations are carried forward to successive stages until they are changed.

Note: It is not permissible to dig to, or below the base of the wall. However, this could be modelled, if required, by specifying a very low bending stiffness for the bottom section of the wall.

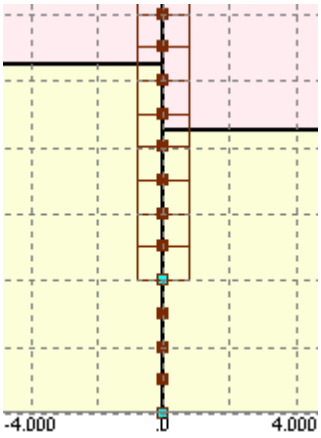
3.5.10 Wall Data

Wall data must be entered in Stage 1 and be specified for all subsequent stages. As stages are added, the previous stage data is copied. The wall can be of uniform or non-uniform stiffness profile which can be changed for each stage. Any revised stiffness will be then be carried forward and used for all subsequent stages, until it is changed again.

Note: Changes in wall stiffness between stages will not adjust the moment curvature relationship that exists at the end of the previous stage. The use of wall relaxation must be used to adjust the moment curvature relationship.

For **automatic node generation**, the top level and bending stiffness of the wall is specified in the Wall Data table. Changes in bending stiffness down the wall can be specified by entering more than one line in this table. The base of the wall will be as manually specified by the user, or as generated by the stability check, as required.

For **manual node generation**, the wall bending stiffness (EI value in kN/m2/m length of wall) is specified at each node location. This allows the stiffness to be varied throughout the length of the wall. The stiffness given to each node applies from that node down to the next node.



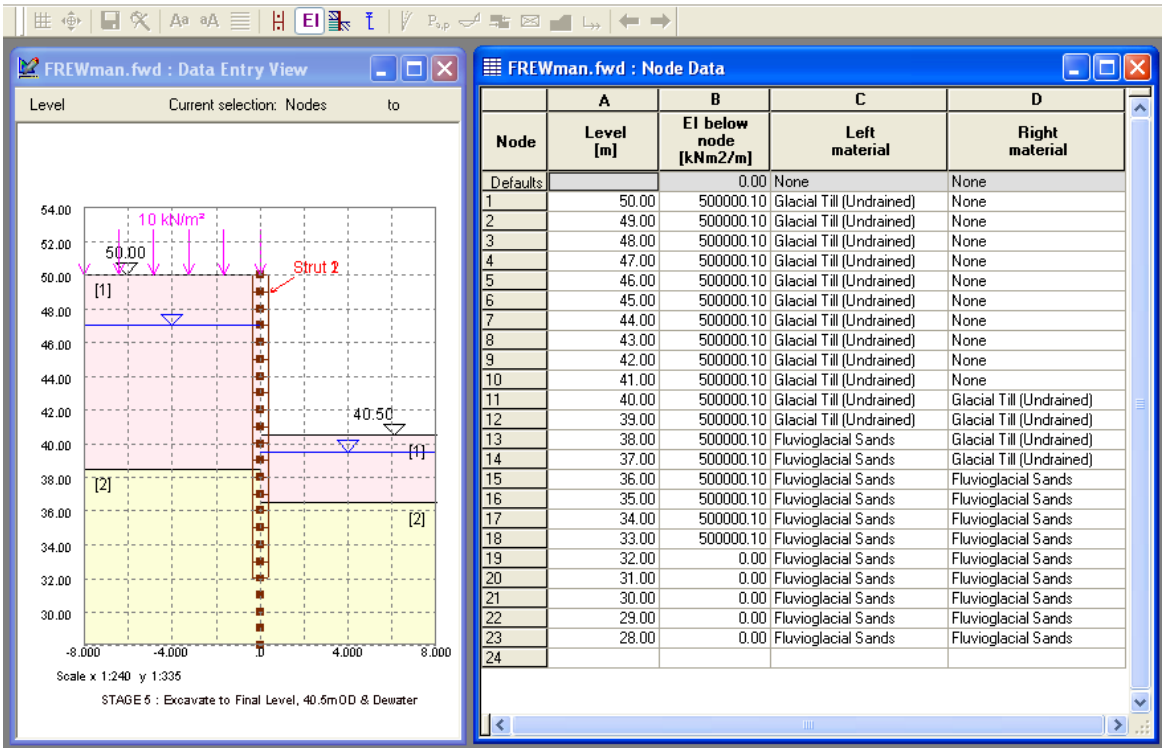
The base of the wall is taken as the first node with a given EI value of zero.

Adding wall stiffness when using manual node generation

To add a wall stiffness profile select the wall stiffness button

on the graphics toolbar or the Wall data option in the 'Stage Operations' window.

© Oasys Ltd. 2014



Node	A Level [m]	B EI below node [kNm2/m]	C Left material	D Right material
Defaults		0.00	None	None
1	50.00	500000.10	Glacial Till (Undrained)	None
2	49.00	500000.10	Glacial Till (Undrained)	None
3	48.00	500000.10	Glacial Till (Undrained)	None
4	47.00	500000.10	Glacial Till (Undrained)	None
5	46.00	500000.10	Glacial Till (Undrained)	None
6	45.00	500000.10	Glacial Till (Undrained)	None
7	44.00	500000.10	Glacial Till (Undrained)	None
8	43.00	500000.10	Glacial Till (Undrained)	None
9	42.00	500000.10	Glacial Till (Undrained)	None
10	41.00	500000.10	Glacial Till (Undrained)	None
11	40.00	500000.10	Glacial Till (Undrained)	Glacial Till (Undrained)
12	39.00	500000.10	Glacial Till (Undrained)	Glacial Till (Undrained)
13	38.00	500000.10	Fluvioglacial Sands	Glacial Till (Undrained)
14	37.00	500000.10	Fluvioglacial Sands	Glacial Till (Undrained)
15	36.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
16	35.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
17	34.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
18	33.00	500000.10	Fluvioglacial Sands	Fluvioglacial Sands
19	32.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
20	31.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
21	30.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
22	29.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
23	28.00	0.00	Fluvioglacial Sands	Fluvioglacial Sands
24				

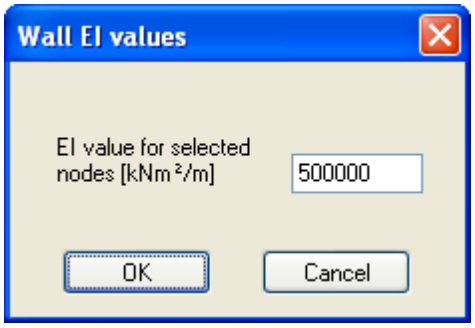
Wall stiffness can either be entered directly into the table or graphically using the procedure given below.

Select the **required range of nodes** on the wall:

1. Place the cursor over the first node and select with the left button.
2. Hold down the Shift key.
3. Place the cursor over the last node (still holding down the Shift key) and select the area of nodes with the left button.

To **designate the wall stiffness**

1. Place the cursor to the left or right of the wall as required.
2. Click the right button on the mouse. This will activate the zones box.



Enter the wall stiffness in the box and select OK.

Editing wall stiffness

© Oasys Ltd. 2014

The wall stiffness can be changed, either

1. by repeating the procedure above to overwrite the entered data or
2. by selecting the required node in the table using the cursor and typing in a new number.

Note : The Wall stiffness extends to the top of the node below. The last node in the added list is therefore not available in the wall window to prevent the wall being extended below the base of the defined problem.

3.5.11 Groundwater

The profile of groundwater can be either hydrostatic or piezometric. These can be different on either side of the wall.

For a **hydrostatic distribution** enter a single piezometer, with zero pressure, at the phreatic surface. The profile of pressure with depth will be linear beneath this level and have a gradient dependent on the specified unit weight of water.

Note: The specified unit weight of water is a single **global value** which is applied to all piezometers on the same side of the wall.

The **piezometric distribution** is specified using a series of pressure heads. The water pressure at any point is computed by interpolating vertically between two adjacent points.

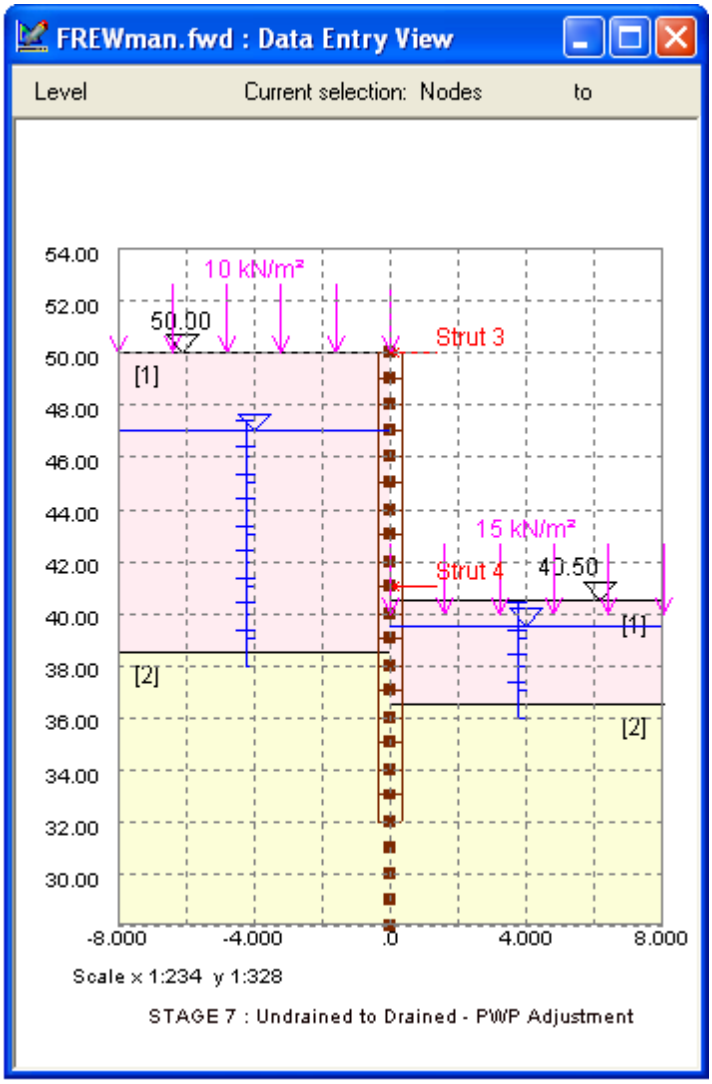
Note: The **highest specified point** must have **zero water pressure**. Negative pore pressures can be specified below the highest point to describe soil suction. The water pressure is assumed to increase hydrostatically below the lowest specified point.

The water pressures are also assumed to be constant laterally from either side of the wall.

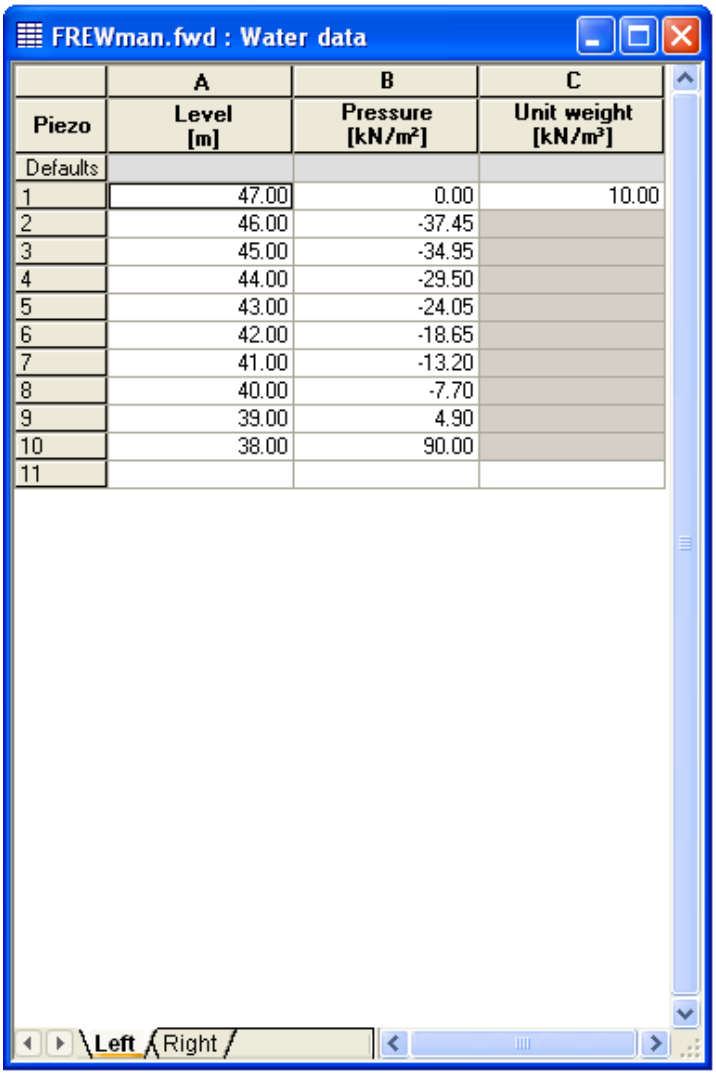
Adding piezometers

Piezometer data is entered by selecting the piezometer button on the graphics toolbar or the in the 'Stage Operations' window. Water data on the left or right side of the wall can be viewed by selecting the appropriate page tab at the bottom of the table.

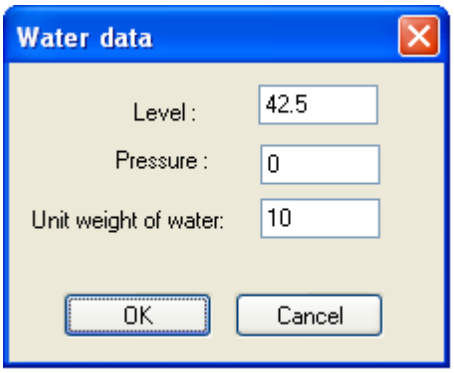
© Oasys Ltd. 2014



© Oasys Ltd. 2014



A piezometric groundwater profile can be entered in the table or by placing the cursor at the appropriate level on the graphical view and clicking the mouse button. This opens the piezometer data box.



This allows the level of the piezometer to be confirmed or edited and the corresponding pressure to be entered. The pressure (P) is given as:

© Oasys Ltd. 2014

$$P = (h_p - h_w)\,\gamma_w$$

where:

h_p = The level of the piezometer

h_w = The level of the piezometric head at the piezometer

γ_w = Global unit weight of water.

Note: Any amendments to the global weight of water will automatically be applied to all piezometers on the same side of the wall.

Editing

Once the information for a piezometer has been entered then the data can be edited or deleted using the tabulated information beside the graphical view.

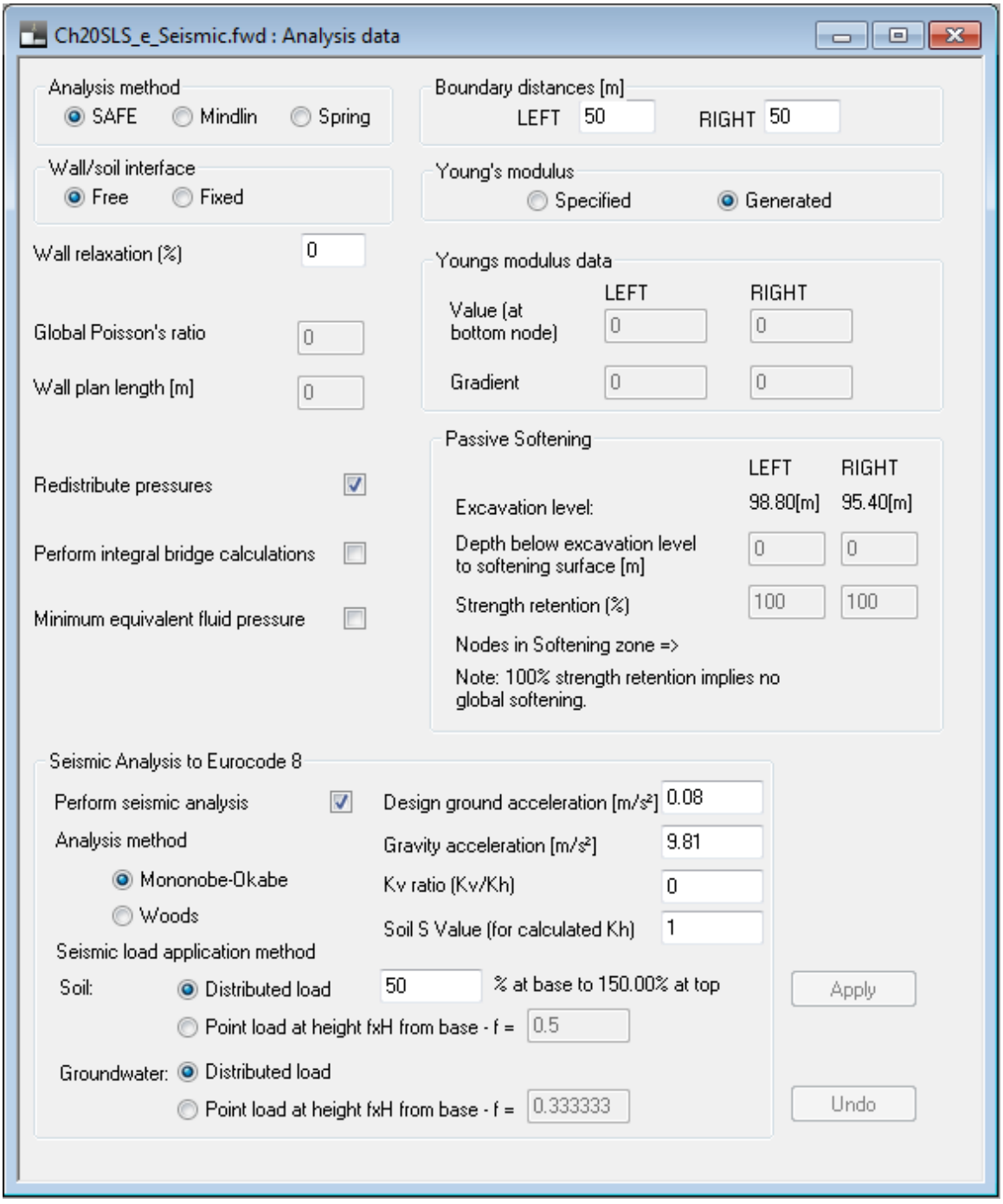
The piezometers can also be deleted by placing the cursor over the location and clicking the left button whilst holding down the Shift key.

3.5.12 Analysis Data

The analysis data dialog allows specification of the following overall parameters required to define the problem.

Note: These parameters can be changed for each stage.

© Oasys Ltd. 2014



3.5.12.1 Model Type

The type of soil model to be used must be selected here. Further data is then requested depending which model is selected.

Safe method	Wall/Soil interface must be selected as 'fixed' or 'free'.

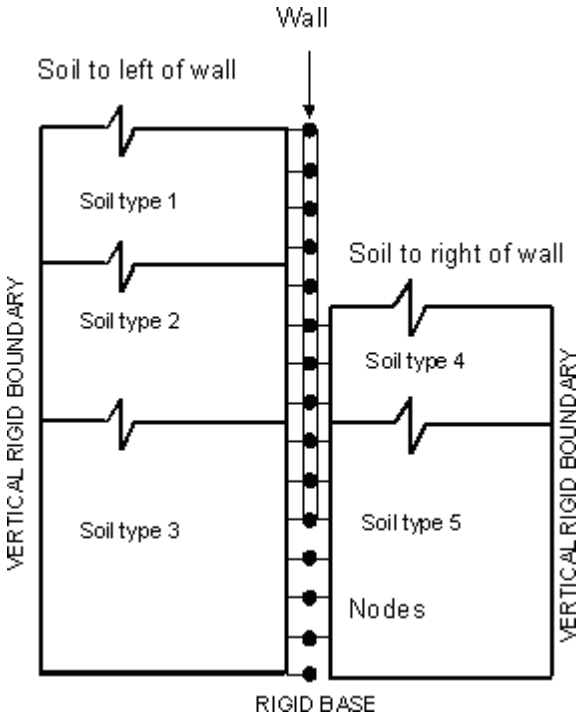
© Oasys Ltd. 2014

Mindlin method	The Global Poisson's ratio and wall plan length must be specified.
Method of Sub-grade Reaction	***Not available yet in windows version***

3.5.12.2 Boundary Distances

When soil stiffness is represented using the "**Safe**" or Mindlin methods, it is assumed that the lowest node specified in the data defines a **horizontal rough rigid boundary**.

For details on inserting Nodes. The rigid boundary should be set at a level where the soil strain, due to excavation or loading is expected to have reduced to near zero.



The distances from the wall to **rigid vertical boundaries** to the LEFT and RIGHT are also required.

For the **Safe** and **Mindlin** methods the vertical boundaries can be used to represent;

1. a plane of reflection in a symmetric excavation
2. or the limit, at some distance from the excavation, beyond which soil strain is expected to have reduced to zero.

Note: The specified distances can be different on either side of the wall.

By restricting the distance at which deformation can occur, the effects of an excavation of limited length can be achieved. For further information see Modelling Axi-symmetric Problems Using Frew.

In the **Sub-grade reaction** method the vertical boundary distances are used to represent the length of the soil spring at each node. Spring lengths may be specified for each node and different values

© Oasys Ltd. 2014

may be given on the left and right sides of the wall.

3.5.12.3 Wall Relaxation

From stage 2 onward it is possible to specify a % relaxation, to model long term behaviour of the wall. For further information, see Creep and Relaxation. The wall relaxation is set to zero in following stages unless changed by the user.

3.5.12.4 Fixed or Free Solution

'Free' and 'fixed' refer to restriction of the vertical displacements of the vertical faces of the left and right soil blocks where they interface with the wall and, below the wall, with each other. This is relevant only to the elastic behaviour of the system, not to the limiting stresses determined by specified values of K_a and K_p.

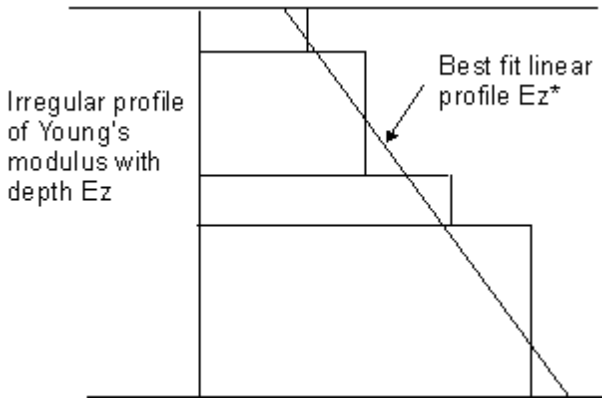
When '**Free**' is used, vertical displacement is permitted, and soil blocks and wall behave as though the interface between them is lubricated, transmitting no shear. When '**Fixed**' is used, the soil blocks are constrained at the interfaces, with no vertical displacement.

Neither the vertical displacements in the 'Free' case nor the shear stresses implied in the 'Fixed' case are explicitly calculated by **Frew**. However, the stiffness matrices, originally set up by **Safe**, are different for the two cases. Simpson (1994) has shown that allowance for vertical shear stresses reduces computed horizontal (elastic) displacements, and this is significant in some cases. Therefore the 'Fixed' case will generally lead to smaller displacements, and users may consider that it is closer to reality since there is generally little vertical displacement on the plane of the wall.

Neither case accurately represents the development of shear stresses on the plane of the wall in response to vertical displacements of the wall and soil, which are largely related to non-elastic movements.

3.5.12.5 Young's Modulus (E)

The value of Young's modulus is given to each soil layer, (see Material Properties). The profile of Young's modulus may therefore vary irregularly with depth.



However, a linear profile is needed in order to use the **Safe** and constant value of E to use the Mindlin method.

© Oasys Ltd. 2014

Safe Model

When using the **Safe** Model **Frew** can **generate** a "best fit" linear profile through the stepped profile created by the individual soil layers on either side of the wall. Alternatively, the user can specify a profile by giving the value of Young's modulus at the lowest node and then a gradient of the line. A positive gradient creates an increasing profile with depth.

When **Frew** generates the best fit line, it then makes further modifications to the stiffness matrix. This attempts to fit the irregular profile of Young's modulus better.
For details on the Accuracy with respect to Young's modulus (E).

Mindlin Model

The Mindlin Model requires a constant value of E. The method can either **generate** a best fit value or use a **user specified** value.

For details on the use of E in the Mindlin method see Approximations used in the Mindlin method.

For both methods, if the "Generate" option is used, an approximate modification is made to allow for the irregular variation of E value with depth. If the "Specified" option is selected then the user must define the required profile.

3.5.12.6 Redistribution of Pressures

The use of **redistribution** can allow for the effects of arching in the soil.

If "no redistribution" is specified, the wall pressures at all points are limited to lie between p_a and p_p. However, if "redistribution" is allowed, it is assumed that arching may take place according to theory presented in Calculation of Active and Passive Limits and Application of Redistribution.

Note: It is considered that the "redistribution" option, while being less conservative is more realistic.

3.5.12.7 Minimum Equivalent Fluid Pressure

If it is required that the total active pressure on the wall at any depth below the ground surface should not drop below a specified value, a Minimum Equivalent Fluid Pressure (MEFP) can be automatically calculated by **Frew**. To use this feature, check the minimum equivalent fluid pressure box on the Analysis Data dialog.

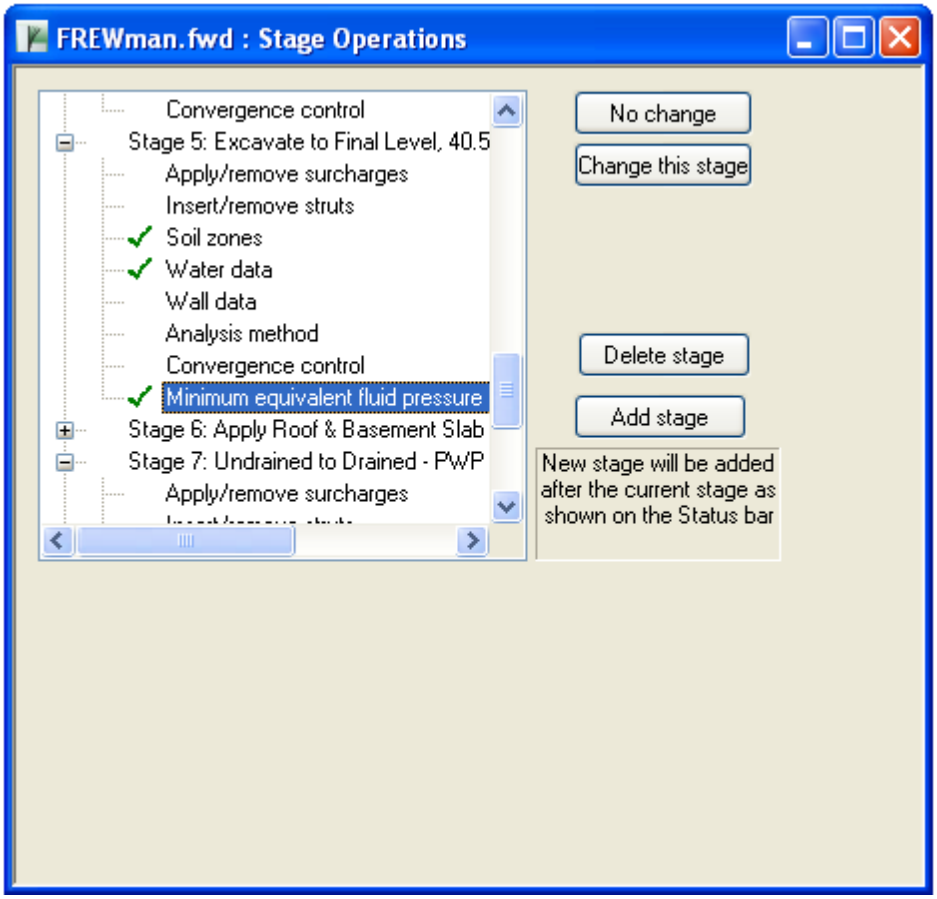
This will add an option to the Stage Operations which allows entry of MEFP parameters.

The MEFP option is available on the Analysis option dialog box.

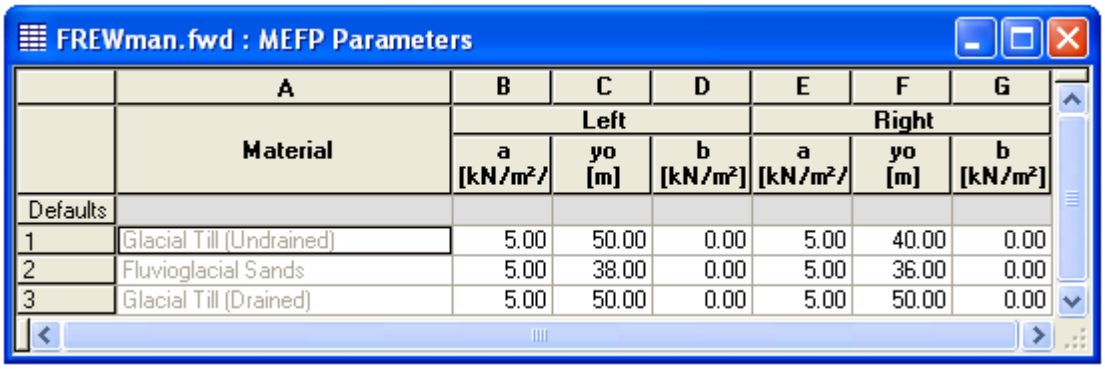
Note: These parameters can be changed for each stage.

If the MEFP is checked then another option is added to the Stage Operations tree view for that stage,called "Minimum Equivalent Fluid Pressure ".

© Oasys Ltd. 2014



Selecting the "Minimum equivalent fluid pressure" option, the MEFP parameters table appears.



	A	B	C	D	E	F	G
	Material	Left			Right		
		a [kN/m²/]	yo [m]	b [kN/m²]	a [kN/m²/]	yo [m]	b [kN/m²]
Defaults							
1	Glacial Till (Undrained)	5.00	50.00	0.00	5.00	40.00	0.00
2	Fluvioglacial Sands	5.00	38.00	0.00	5.00	36.00	0.00
3	Glacial Till (Drained)	5.00	50.00	0.00	5.00	50.00	0.00

This table has a record for each material with parameters for left and right sides.

The required MEFP is specified as a linear relationship with depth plus an optional constant value.

a gradient of pressure (default 5 kPa/m)
$\mathbf{y_0}$ level (default is top level of material)
b constant value (default 0)

Note: The user should really set the values to zero where they judge they are not needed or would

© Oasys Ltd. 2014

incorrectly affect the results, e.g. on the passive side of the wall.

During analysis, the active pressure is set to a minimum of $a*(y_0-y)+b$, where y is the level of the node. If this is the governing criterion on active pressure, an 'm' symbol is shown in the tabular output. In the graphical output, the MEFP-derived pressure is just plotted as part of the normal active pressure line.

3.5.12.8 Passive Softening

This analysis option can be specified for individual stages. If passive softening is used, then the undrained shear strength of the material is assumed to increase linearly from zero to the global softening value (%) over the passive softening depth entered.

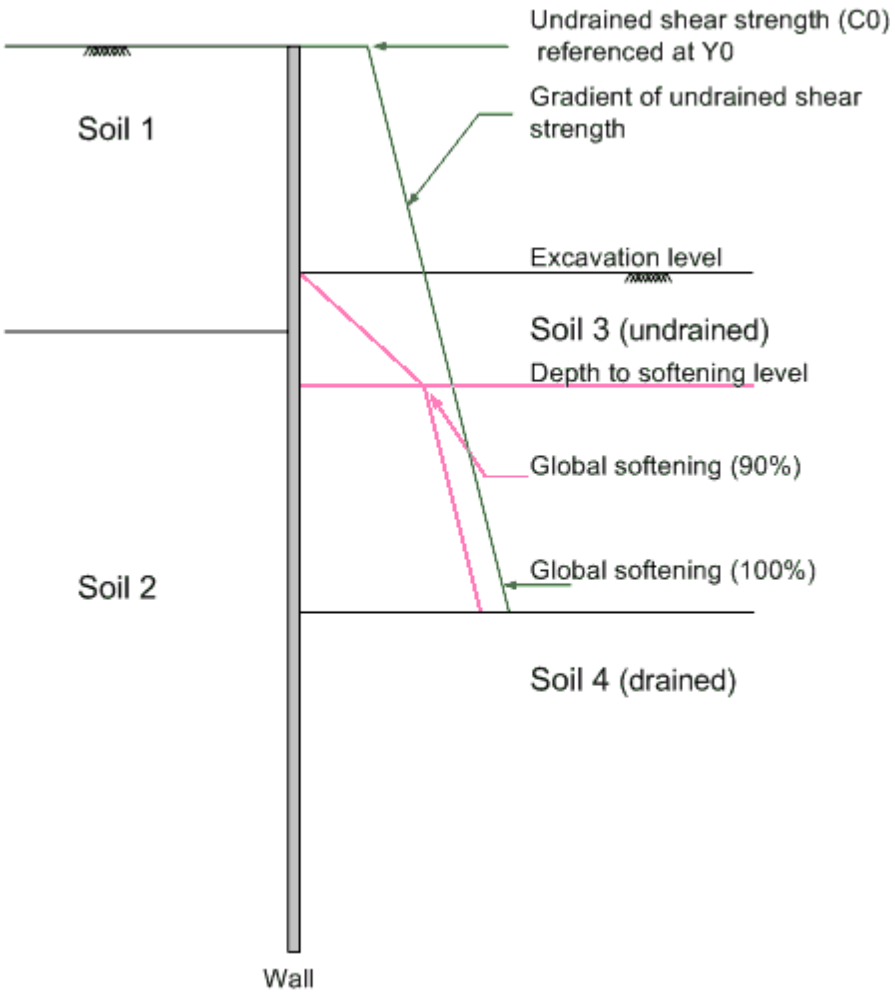
To use this option enter:

- The depth to which surface softening occurs, (distance units)
- The magnitude of original strength/stiffness to be used for global softening, (%)

The strength of all undrained material below a softening depth will be reduced to global softening (%) of the value in the Material Properties table.

Note: Since either (or both) sides of the wall can be excavated, the user can specify separate parameters for both sides of the wall.

© Oasys Ltd. 2014

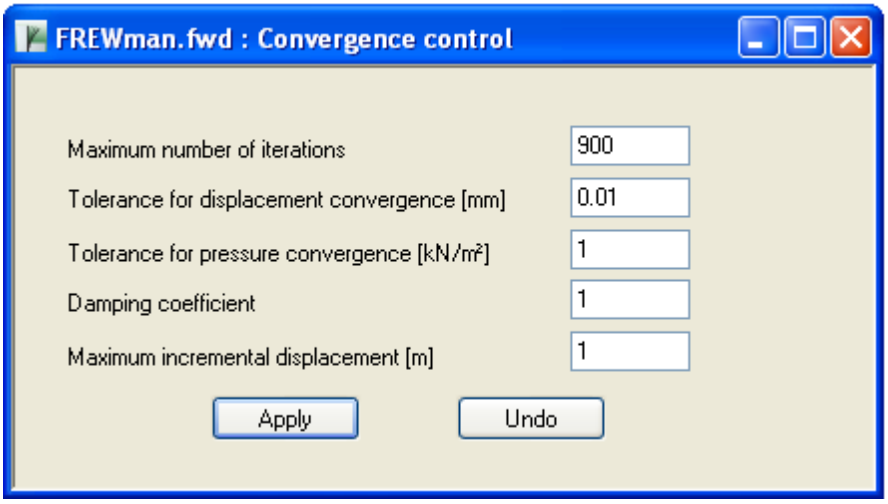


3.5.13 Convergence Control

The following parameters are required to specify the convergence criteria for the calculations.

Note: These parameters can be changed for each stage.

© Oasys Ltd. 2014



Convergence control parameters may be varied from the default values offered to improve the speed/ accuracy of the solution, or to reduce the chance of numerical instability.

3.5.13.1 Maximum number of Iterations

The maximum number of iterations for each stage can be specified. The stage calculations will complete at this maximum number of iterations if this is reached before both the tolerance criteria given for the displacement and pressure are satisfied.

If the tolerance levels are reached first then the stage calculations will also complete.

Note: The default value for the maximum number of iterations is given as 900.

3.5.13.2 Tolerance for Displacement

The maximum change of displacement between successive iterations. The absolute error in the result will be considerably larger (typically by a factor of 10 to 100). The default value is 0.01mm.

3.5.13.3 Tolerance for Pressure

The maximum error in pressure (i.e. how much the pressure at any node is below the active limit or in excess of the passive limit. This is an absolute value and the default value is 0.1 kPa.

3.5.13.4 Damping Coefficient

The damping coefficient used in the analysis. If convergence is slow this can be increased. If instability is apparent it may possibly be solved by reducing this. The default value is 1.0.

© Oasys Ltd. 2014

3.5.13.5 Maximum Incremental Displacement

Maximum deflection in one stage. The default value is 1.0m

4 Frew-Safe Link

Frew analyses the soil structure interaction of the retaining wall. Frew calculates the pressures, displacements etc. at the wall. However, if one is interested in the movements of soil beneath the wall, Frew will not be adequate. However, the same problem can be modeled in Safe. The Frew-Safe link feature enables the user to create a Safe model which is nearly equivalent to the Frew model. This feature involves creation of a Gwa file. The Gwa file format is a text format used by Oasys Gsa to transfer data across different programs.

The following steps are involved in the process:
1. Validation of Frew data.
2. Entry of wall data.
3. Export of data from Frew file to Gwa file.
4. Import of data from Gwa file to Safe file.

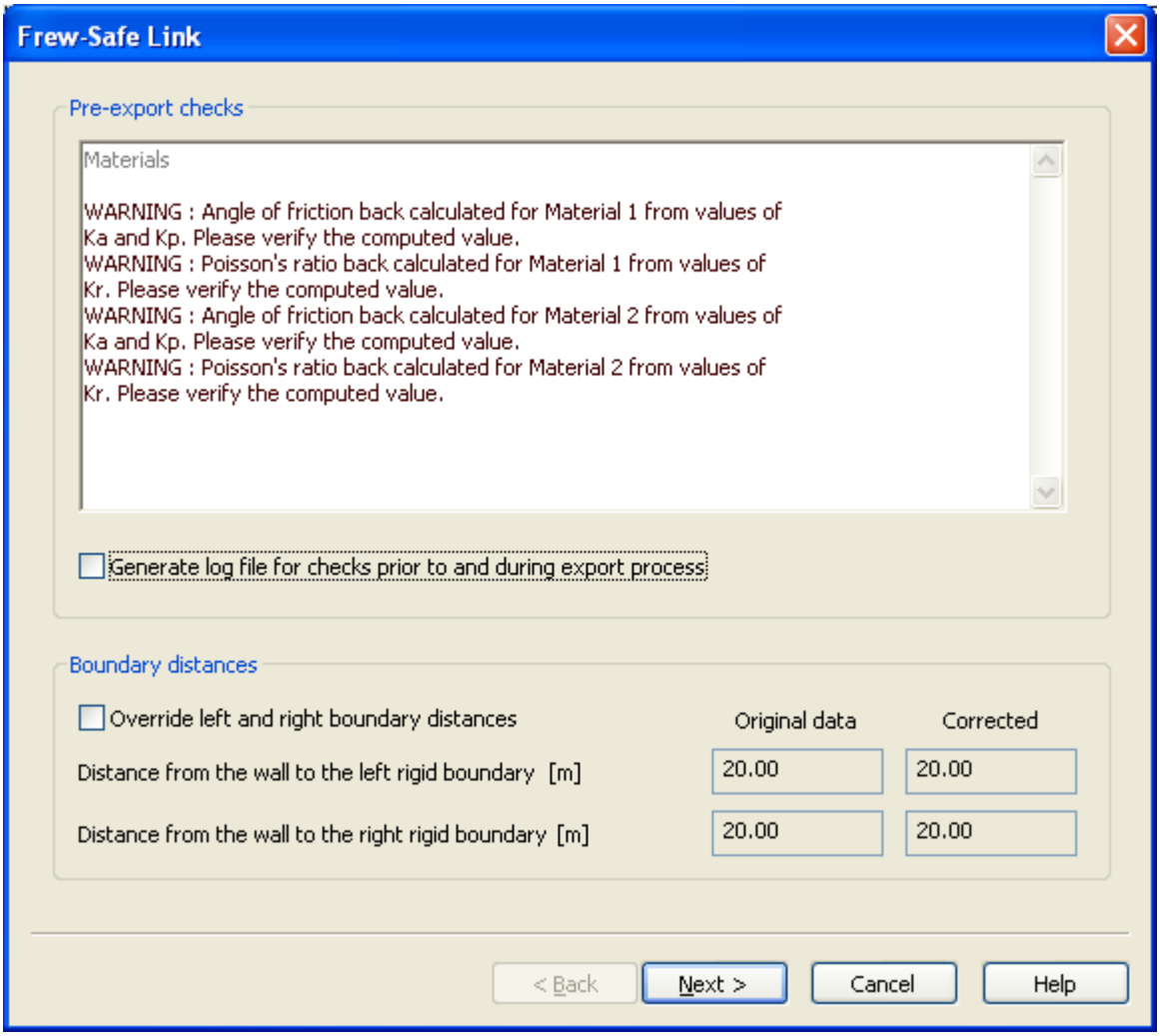
4.1 Data Entry

The Frew-Safe Link wizard is invoked by clicking on the "Export To Safe" button in the file menu.

On clicking the above mentioned button, a file save dialog opens up and prompts for the name of a Gwa file to save the data.

After the user specifies the file name, the existing Frew data is validated. If there are any warnings or errors, they are displayed in the wizard. Warnings can be ignored, but the data cannot be exported if there are any errors. If the relevant checkbox on this page is checked, a log file which contains all the errors and warnings during the export process will be created.

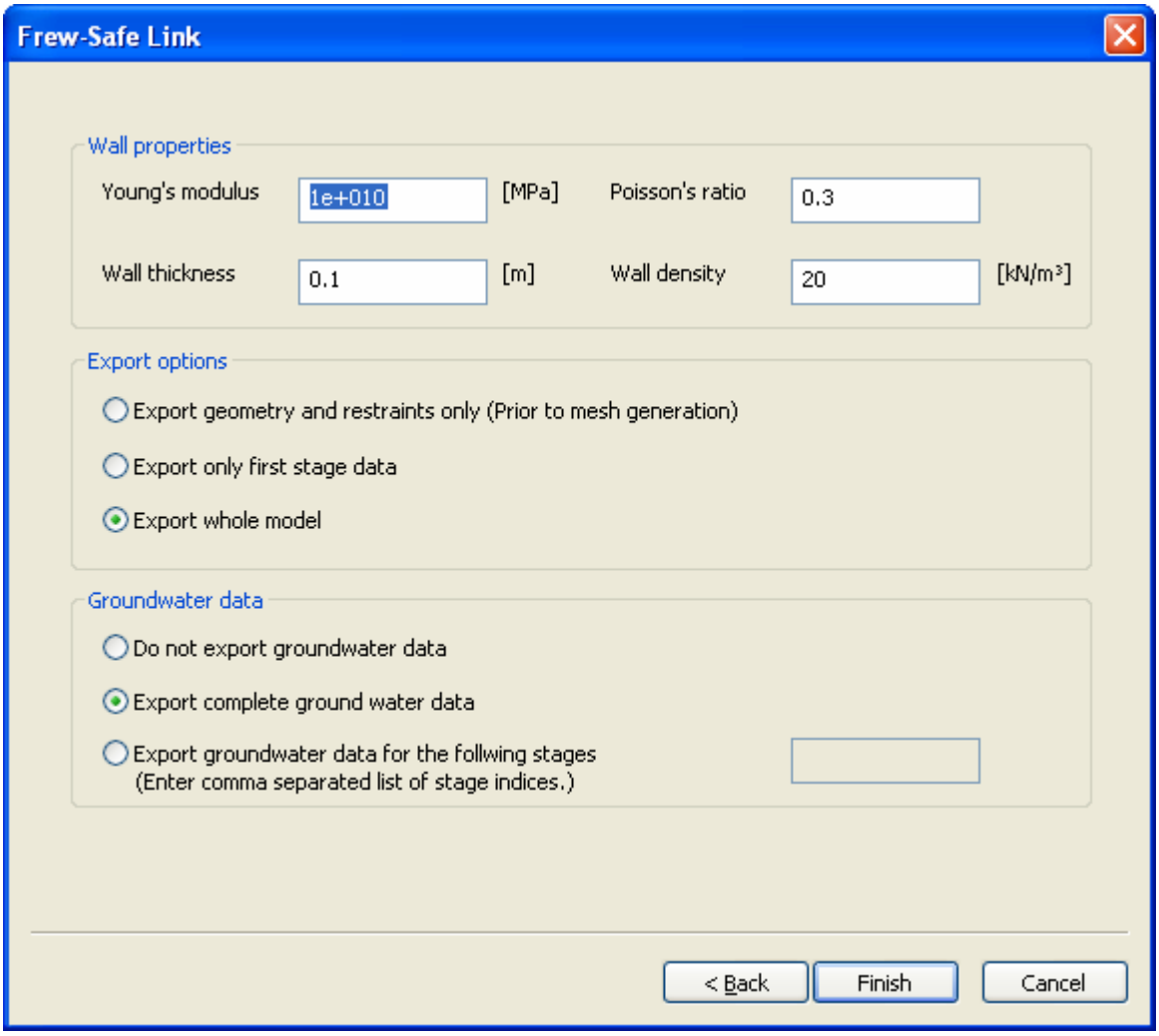
© Oasys Ltd. 2014


Frew-Safe Link
Pre-export checks
Materials
WARNING : Angle of friction back calculated for Material 1 from values of Ka and Kp. Please verify the computed value.
WARNING : Poisson's ratio back calculated for Material 1 from values of Kr. Please verify the computed value.
WARNING : Angle of friction back calculated for Material 2 from values of Ka and Kp. Please verify the computed value.
WARNING : Poisson's ratio back calculated for Material 2 from values of Kr. Please verify the computed value.
Generate log file for checks prior to and during export process
Boundary distances
Override left and right boundary distances
Original data
Corrected
Distance from the wall to the left rigid boundary [m]
20.00
20.00
Distance from the wall to the right rigid boundary [m]
20.00
20.00
< Back
Next >
Cancel
Help

The user may also specify different boundary distances, than those input in the actual Frew file. This is particularly useful in cases when large vertical boundary distances from wall have been specified.For further information see Accuracy of modelling boundaries in Frew.

If there are no errors, the "Next" button will open the next page of the wizard, where the wall data should be entered. This includes the wall thickness, density, Young's modulus and Poisson's ratio for the wall material.

© Oasys Ltd. 2014



The following three options are provided for exporting data:

1. Export geometry and restraints only - If this option is selected, only the critical points, lines and areas are exported, together with the boundary conditions at the ends of the model. No mesh generation data is exported. Thus, data pertaining to surcharges, struts, etc. which are dependent on the mesh data, are also not exported. This option is useful if the user wants to add additional geometric entities in the model after importing into Safe data file.

2. Export only first stage data - This option enables the user to export the entire first stage data, including mesh generation data, surcharges, struts etc. to Safe. This is useful if the user is interested in replicating the same geometry and only the initial conditions of the Frew model in the Safe model.

3. Export whole model - If the user selects this option, almost the whole data, barring some unsupported features which will be detailed later, will be exported to Safe.

The user is further provided three choices to export groundwater data:

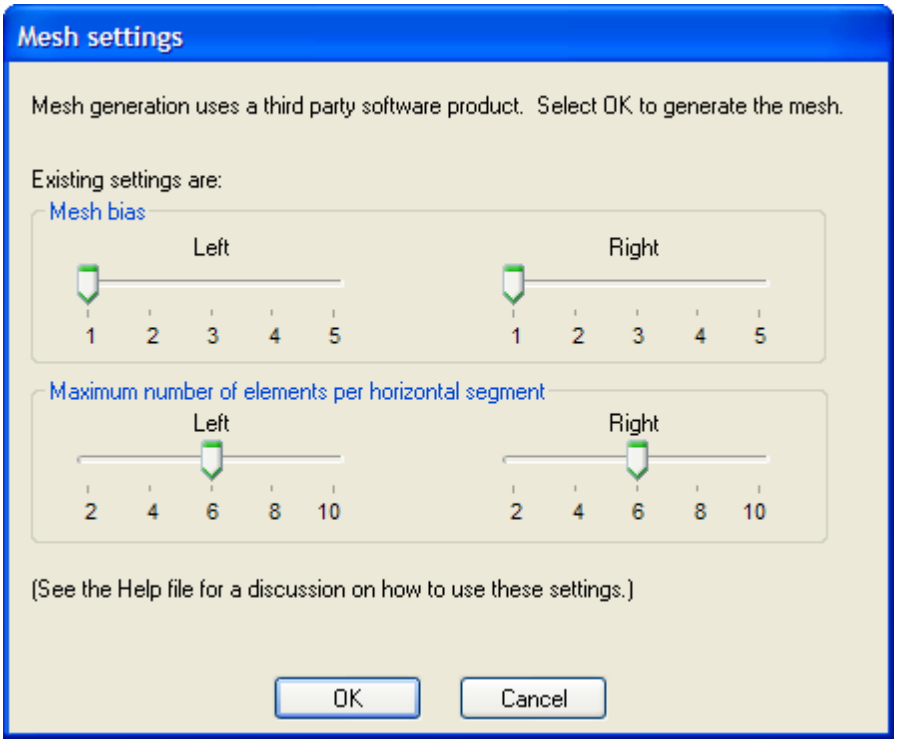
1. Export no groundwater data - In this case, groundwater data is completely suppressed during the export process.

© Oasys Ltd. 2014

2. Export complete groundwater data - In this case, the whole groundwater data from all the stages is exported.

3. Export selective stages- If this option is chosen, the user has to specify the comma separated list of stages for which the groundwater data must be exported. This option may be used for excluding stages involving transition from undrained behaviour to drained behaviour. In Frew, the user has to calculate these transient pore pressures himself. However, in Safe different approaches can be adopted for calculating the transient pore pressures. There may be cases when the user calculate transient pore pressure data in Frew may not accurately model the problem in Safe. In such situations, the user may want to filter out groundwater data from certain stages.

On clicking "Finish", the following "Mesh Settings" dialog pops up if the user chooses "Export only first stage data" or "Export whole model" options:

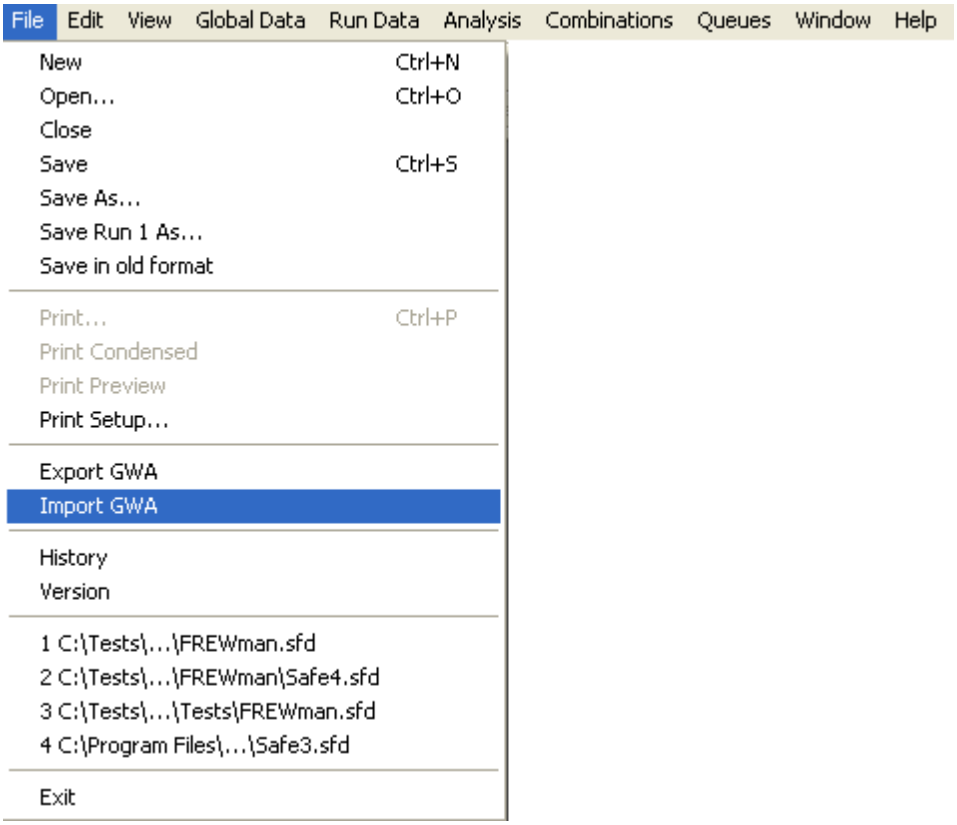

Mesh settings
Mesh generation uses a third party software product. Select OK to generate the mesh.
Existing settings are:
Mesh bias
Left
Right
1 2 3 4 5
1 2 3 4 5
Maximum number of elements per horizontal segment
Left
Right
2 4 6 8 10
2 4 6 8 10
(See the Help file for a discussion on how to use these settings.)
OK
Cancel

This dialog allows the user to bias the mesh along horizontal segments as desired. The biased segments have more nodes towards the wall. The horizontal segments correspond to horizontal lines running from the left boundary to right boundary. A horizontal segment typically joins points located at the boundary with a point on the wall or any surcharge points, two surcharge points etc.

The user can also specify the maximum number of elements that can be generated along a horizontal segment. This option may help the user to increase the number of elements if necessary. The default value is 6. However, this does not affect the number of elements generated along the wall width, which is always 2.

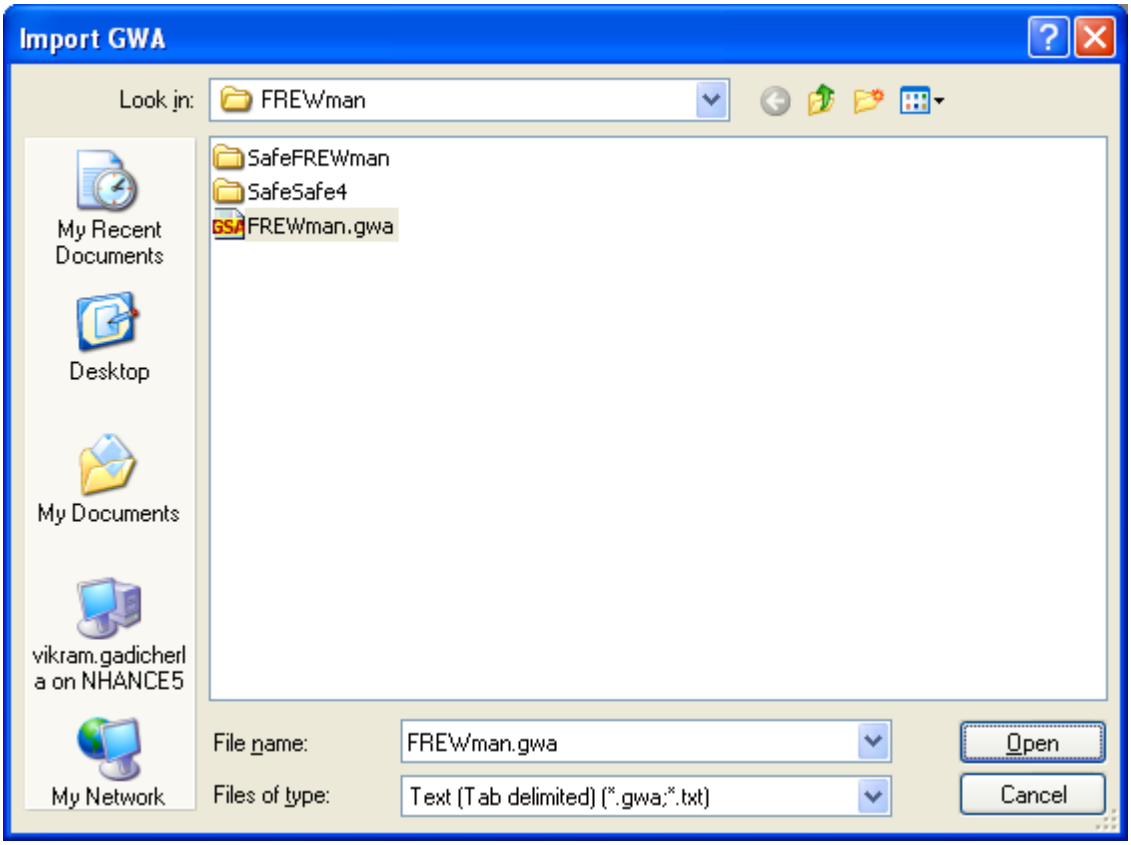
Then, the required Gwa file is created. The data from this file can be imported into the Safe file by clicking "Import Gwa" menu button in the **Safe** program.

© Oasys Ltd. 2014



The user is then prompted to choose the required Gwa file. Upon selection, the data is transferred from the Gwa file to Safe file.

© Oasys Ltd. 2014



4.2 Data Conversion

As mentioned before, this feature enables the user to create the equivalent Safe model from Frew model. There are some approximations and limitations in this data conversion from Frew to Safe. The following topics detail the assumptions, limitations and approximations involved in this process.

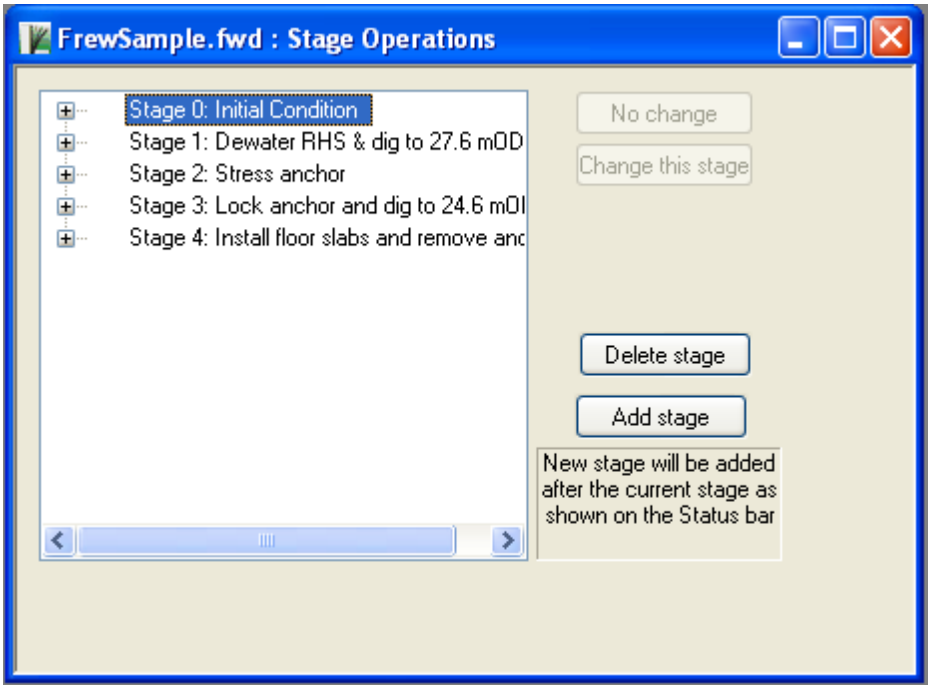
Stages/Runs
Geometry
Restraints
Surcharges
Struts
Materials
Groundwater
Unsupported Features

4.2.1 Stages/Runs

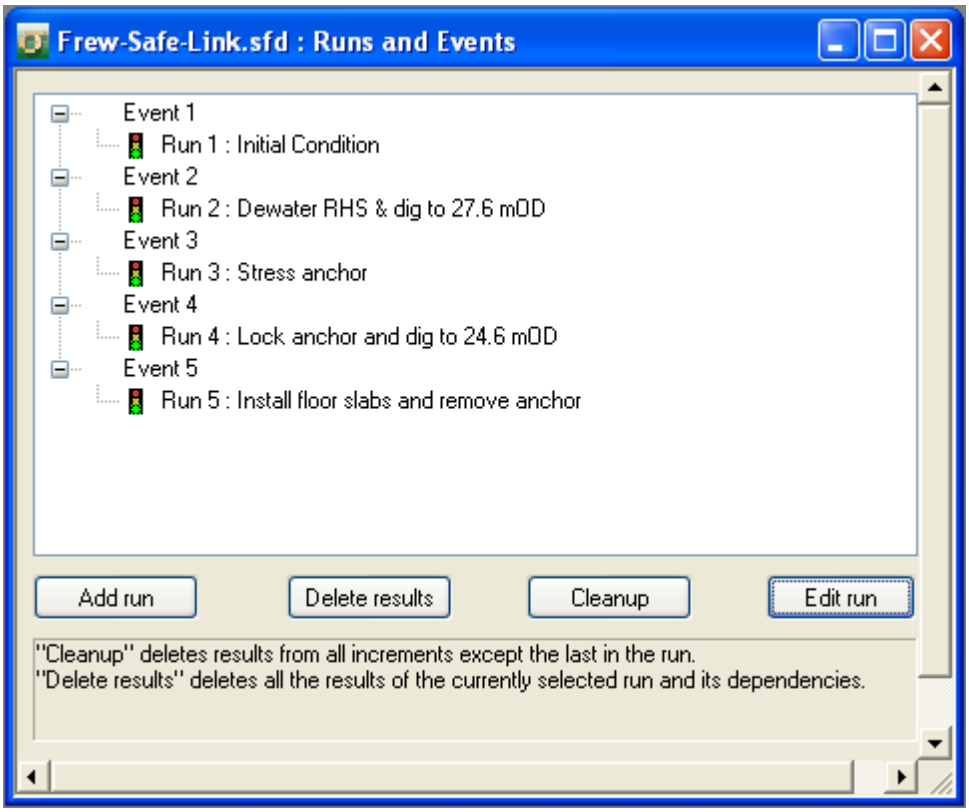
Stages in Frew are roughly equivalent to Runs in Safe. All stages in a Frew model translate to a sequence of runs following each other in Safe, without any branching.

Frew Stages:

© Oasys Ltd. 2014


FrewSample.fwd : Stage Operations
Stage 0: Initial Condition
Stage 1: Dewater RHS & dig to 27.6 mOD
Stage 2: Stress anchor
Stage 3: Lock anchor and dig to 24.6 mOI
Stage 4: Install floor slabs and remove anc
No change
Change this stage
Delete stage
Add stage
New stage will be added after the current stage as shown on the Status bar

Safe Runs:


Frew-Safe-Link.sfd : Runs and Events
Event 1
Run 1 : Initial Condition
Event 2
Run 2 : Dewater RHS & dig to 27.6 mOD
Event 3
Run 3 : Stress anchor
Event 4
Run 4 : Lock anchor and dig to 24.6 mOD
Event 5
Run 5 : Install floor slabs and remove anchor
Add run
Delete results
Cleanup
Edit run
"Cleanup" deletes results from all increments except the last in the run.
"Delete results" deletes all the results of the currently selected run and its dependencies.

© Oasys Ltd. 2014

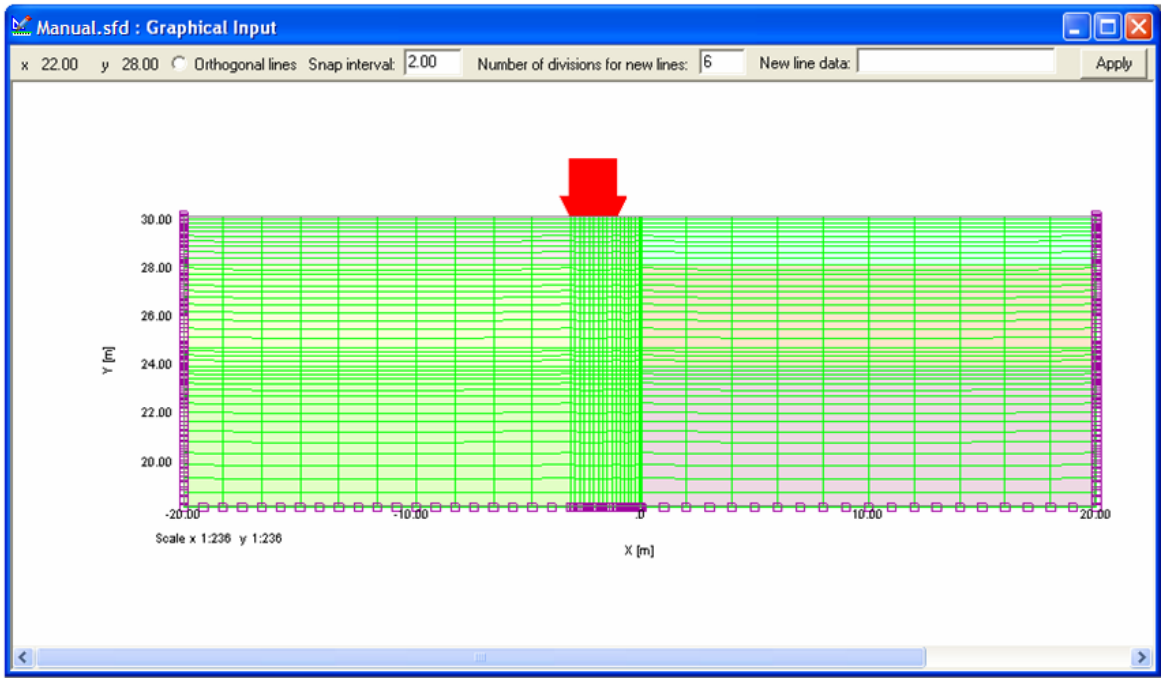
4.2.2 Geometry

For a given Frew model, points are generated at locations corresponding to:

- Material layer boundaries
- Groundwater levels
- Surcharge levels
- Strut locations
- Wall end points.
- Rigid boundary intersections

A series of lines connect these points in the form of a grid. Areas are formed from these lines.

Once this geometry is created, mesh generator is called if required by the user.The node spacing is dense towards excavation levels. Unlike in Frew, the wall in Safe is made up of Quad-8 elements.



4.2.3 Restraints

The following points should be noted regarding the export of data related to restraints:

1. All restraints in the model are of pin-type, and are applied at the rigid boundaries.
2. The restraints are constant across all stages. Variable boundary distances across different stages not supported.

© Oasys Ltd. 2014

4.2.4 Surcharges

The surcharges are applied as element edge loads in the corresponding stages.

Frew surcharge data:

Manual.fwd : Surcharges

	A	B	C	D	E	F	G	H	I
Surcharge	Stage		Side	Level [m]	Type	Pressure [kN/m²]	Offset [m]	Width [m]	Ks
	In	Out							
Defaults			Left		UDL				
1	0		Left	30.00	Strip	25.00	1.00	2.00	1.00
2									

Equivalent Safe element load data:

Safe1 : Pressure Loads

	A	B	C	D
No.	List of nodes	Direction	Pressure [kN/m²]	
			1st node	3rd node
Defaults		Normal		
1	3 581 66	Normal	25.00	25.00
2	66 585 2	Normal	25.00	25.00
3				

4.2.5 Struts

Struts in Frew modeled as springs in Safe, and act at nodes located on the wall axis.

Lever arm information is **not** exported from Frew to Safe.

Pre-stress is applied as a node load in Safe. The prestress along the spring axis direction is resolved into components along the X and Y axes, and applied as a pair of node loads in Safe.

Frew strut data:

Frew-Manual.fwd : Struts

	A	B	C	D	E	F	G
Strut	Stage		Node number	Prestress [kN/m]	Stiffness [kN/m/m]	Angle [°]	Lever arm [m]
	In	Out					
Defaults							
1	2	3	3	200.00	0.00	40.00	0.30
2	3	4	3	200.00	5000.00	40.00	0.30
3	4		1	0.00	30000.00	0.00	0.00
4	4		7	0.00	30000.00	0.00	0.00
5							

Equivalent Safe strut data:

© Oasys Ltd. 2014

Safe1 : Nodal Springs

Spring No.	A Node	B x/y inclination from horizontal [°]	C Spring constants [kN/m per m] x/y
Defaults	0	0	0
1	43	40	0
2	43	40	5000
3	44	0	30000
4	45	0	30000
5			

Safe1 : Node Loads

No.	A Node(s)	B Direction	C Load [kN/m]
Defaults		Y	
1	43	X	153.2
2	43	Y	128.6
3			

4.2.6 Materials

In Frew, material data is specified only for Soil strata. In the Safe model, the excavations are represented using "void" material, the wall is modeled using a linear elastic material, and the soil is modeled using Mohr-Coulomb materials.

Safe void material data:

Safe1 : Water or air properties

	A Zone:Description	B E1 [kPa]	C NU13	D E2 [kPa]	E NU12	F G12 [kPa]	G Alpha1 [°]	H Ta [kPa]
Defaults								
1	3: Void	1e-005	0				0	0

The wall material data is supplied by the user during the export process.

© Oasys Ltd. 2014

Safe1 : Linear elastic material properties

	A	B	C	D	E	F	G
	Zone:Description	E1 [kPa]	NU13	E2 [kPa]	NU12	G12 [kPa]	Alpha1 [*]
Defaults							
1	4: Wall	1e+007	0.3				0

The Frew material model has all the data required for Safe Mohr-Coulomb model, except for the angle of friction φ, and Poisson's ratio ν.

Frew material data:

Frew-Manual.fwd : Materials

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U
Material	Description	E0 [kN/m²]	Unit weight [kN/m³]	Ko	Earth pressure coefficient	Phi (Ø) [*]	Delta / phi ratio	Beta (ß) [*]	Cw / c ratio	Ka	Kp	Kac	Kpc	Kr	c0 [kN/m²]	y0 [m]	Gradients [kN/m²/m]		Drained/ undrained	Shape factor	Material no. for effective stress parameters
																	c	E			
Defaults					Calculated														Drained		
1	Material 1	50000	20.0	0.60	User Specified					0.20	6.00	0.894	4.899	0.2	0.00	0.0	0.00	0.00	Drained		
2	Material 2	25000	19.0	0.80	User Specified					0.35	3.50	1.183	3.742	0.3	10.00	0.0	0.00	0.00	Drained		
3																					

Equivalent Safe Mohr-Coulomb material data:

Safe1 : Elastic Mohr/Coulomb material properties

	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O
	Zone:Description	E1 [kPa]	NU13	E2 [kPa]	NU12	G12 [kPa]	Alpha1 [*]	c [kPa]	Phi [*]	Tf	Yo [m]	Beta	Dilation	Gradient of c [kPa/m]	Strength Increase
Defaults															0
1	1: Material 1	5e+004	0.1667				0	0	43.7	0.01	0	0	0	0	0
2	2: Material 2	2.5e+004	0.2308				0	10	31.27	0.01	0	0	0	0	0

Poisson's ratio is calculated from K_r using the following equation:

$$K_r = \frac{\nu}{1-\nu}$$

Angle of friction is the average of values obtained from the following expressions for coefficients of active and passive pressure:

$$K_a = \frac{1-\sin\phi}{1+\sin\varphi}$$

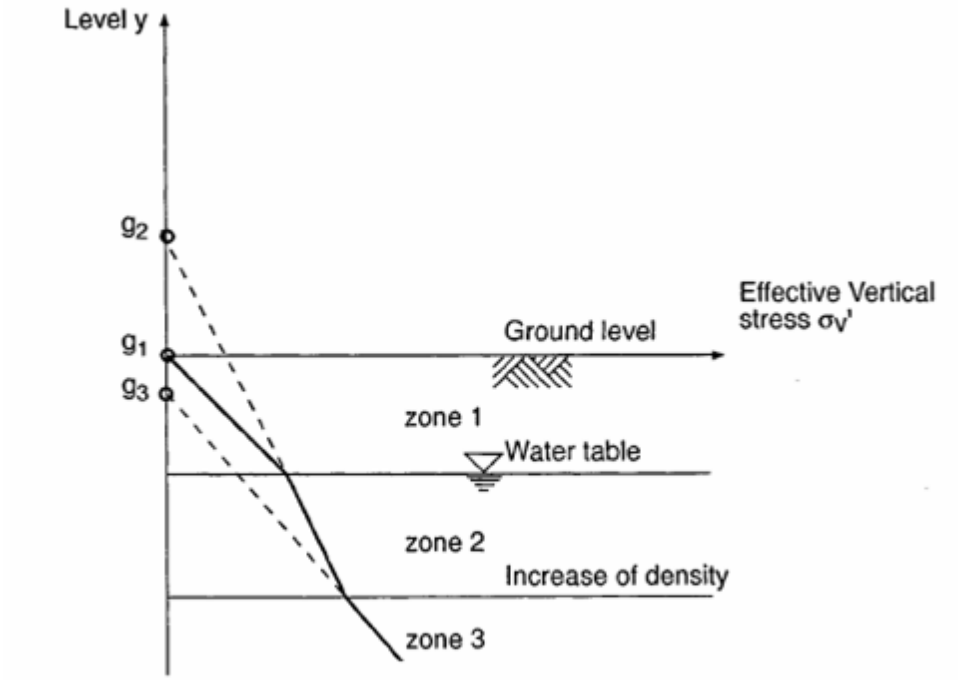
$$K_p = \frac{1+\sin\phi}{1-\sin\varphi}$$

The angle of friction is limited to a maximum value of about 50 degrees.

© Oasys Ltd. 2014

4.2.6.1 First Stage Material

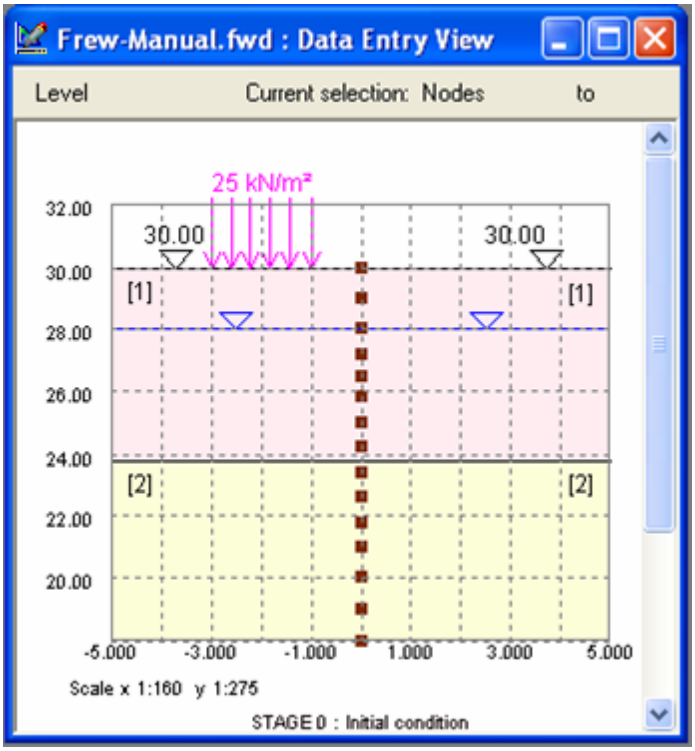
In Safe, as discussed earlier, the initial ground stresses are calculated using two parameters, K0 and 'g'. The slope of the effective vertical soil stress profile is the effective unit weight of the material. K0 determines the slope of the effective horizontal stress profile, and 'g' is the reference level for the linear stress distribution, i.e. the level at which the vertical or horizontal stresses are zero(see figure below).



When a soil stratum is partially submerged, the 'g' parameter differs for the wet and dry part of the stratum, even though all other parameters are identical. Hence two materials are needed to model the partially submerged material in the first stage. These extra materials are generated and the appropriate 'g' values for all the strata are calculated during the export process. The following figures illustrate this situation:

Frew first stage material data:

© Oasys Ltd. 2014



Frew-Manual.fwd : Materials

	A	B	C	D	E	F	G	H	I	J
Material	Description	E0 [kN/m²]	Unit weight [kN/m³]	Ko	Earth pressure coefficient	Phi (Ø) [°]	Delta / phi ratio	Beta (ß) [°]	Cw / c ratio	Ka
Defaults					Calculated					
1	Material 1	50000	20.0	0.60	User Specified					0.20
2	Material 2	25000	19.0	0.80	User Specified					0.35
3										

Equivalent Safe first stage material data:

Safe1 : General material properties

	A	B	C	D	E	F	G	
	Description	Type	Deform. mode	Drain. condn.	Total weight density [kN/m³]	Init Ko	Param 'g'	Wt d of po [kN
Defaults	Material #	Linear Elastic	PLSTRN	No water				
1	First Stage Material 1	Elastic Mohr/Coulomb	PLSTRN	Drained	20	0.6	30	
2	First Stage Material 2	Elastic Mohr/Coulomb	PLSTRN	Drained	20	0.6	32	
3	First Stage Material 3	Elastic Mohr/Coulomb	PLSTRN	Drained	19	0.8	37.58	
4	First Stage Material 4	Elastic Mohr/Coulomb	PLSTRN	Drained	20	0.6	30	
5	First Stage Material 5	Elastic Mohr/Coulomb	PLSTRN	Drained	20	0.6	32	
6	First Stage Material 6	Elastic Mohr/Coulomb	PLSTRN	Drained	19	0.8	37.58	
7	Void	Water or Void	PLSTRN	No water	1e-005	0	0	
8	Wall	Linear Elastic	PLSTRN	No water	0.02	0.5	0	
9								

© Oasys Ltd. 2014

4.2.7 Groundwater

Frew and Safe use different approaches for modeling pore pressure distribution. Following are a couple of important differences.

Frew	Safe
Piecewise linear interpolation of pore pressure.	Radius of influence approach.
All soil zone materials share the same pore pressure distribution data.	Each material has its own pore pressure distribution

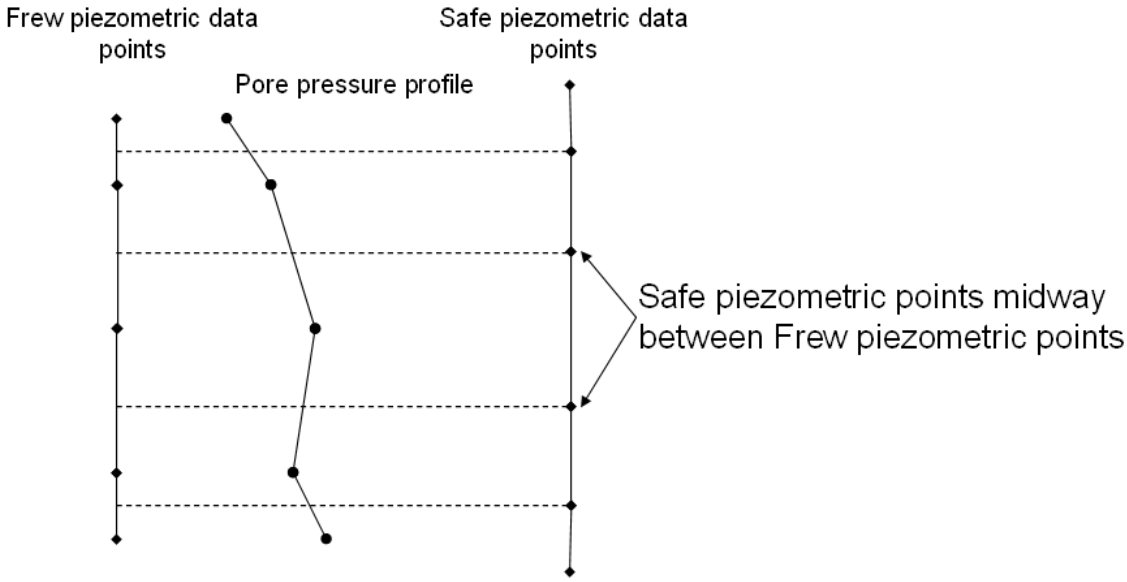
In Safe, each data point is characterized by pore pressure value, its gradient, and a radius of influence. The net pore pressure at a given point is the weighted average of the pore pressures calculated at the given point using the existing pore pressure data points.

The weights for points which lie within a square defined by this radius of influence, the weights are typically much higher compared to the weights for the points located outside the square.

In Frew, we can have different pore pressure gradients above and below a particular pore pressure data point. However, this situation is not possible in Safe for a given data point, as only a single pore pressure gradient is specified.

In order to generate a roughly equivalent pore pressure distribution in Safe, the pore pressures are calculated at locations midway between the Frew pore pressure data points.

© Oasys Ltd. 2014


Frew piezometric data points
Safe piezometric data points
Pore pressure profile
Safe piezometric points midway between Frew piezometric points

4.2.8 Unsupported Features

The following features are currently not supported by the Frew-Safe Link feature:

- Free soil-wall interaction
- Passive softening
- Generated Young's modulus profiles
- Minimum equivalent fluid pressure
- Wall relaxation
- Mindlin model
- Sub-grade reaction model.
- Seismic assessment.

5 Integral Bridge Analysis

Frew can be used to perform integral bridge analysis. This analysis is based on PD 6694-1. In this analysis, strut loads representing the expansion and contraction of the bridge are applied in consecutive stages. New stages corresponding to deck contraction in winter and deck expansion in summer are added as the analysis proceeds.The analysis continues until the convergence in the d/H values at the front and back of the wall are achieved.

© Oasys Ltd. 2014

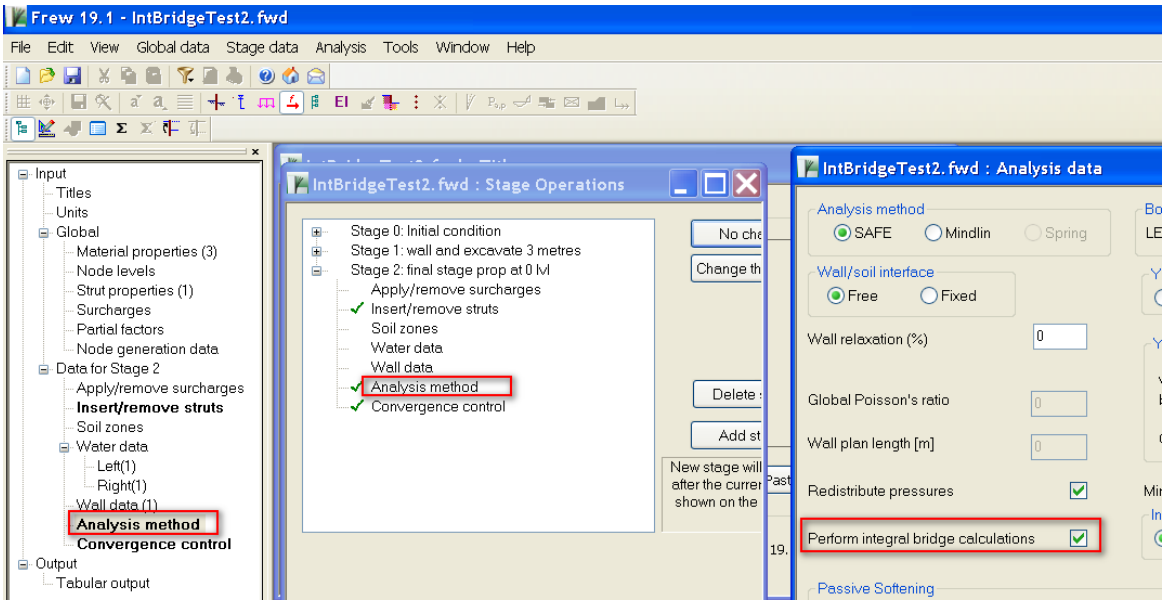
5.1 Data Entry

To use the integral bridges analysis feature, the following steps are required:

1. All the stages up to integral bridge analysis should be defined as before i.e. if there are 3 stages before initial winter contraction, these 3 stages should be defined as usual.

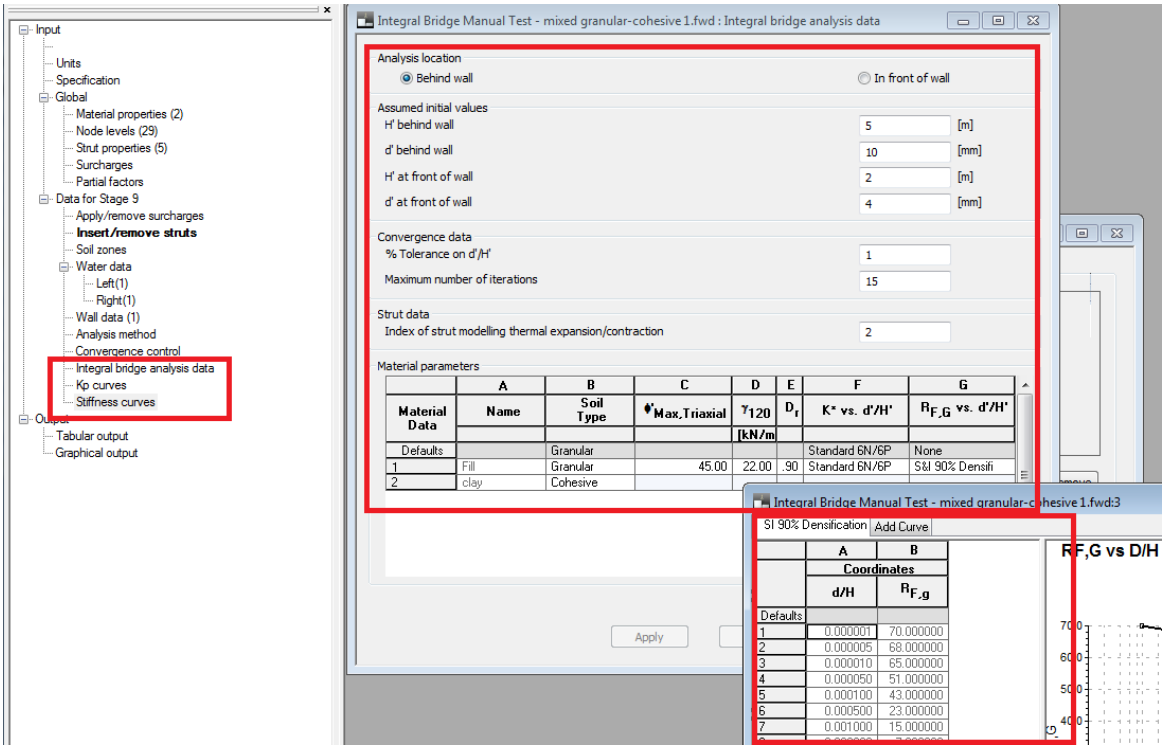
2. The user should only define one stage as the integral bridge analysis stage, and this should be the last stage. This is to be defined by the user at the end of all non-integral bridge stages.

3. The integral bridge stage can be added in the same way as the normal stage. However, to specify a particular stage as integral bridge analysis stage, the user should open the "Analysis Options" dialog for a particular stage, and check the "Perform Integral bridge calculations" check box.



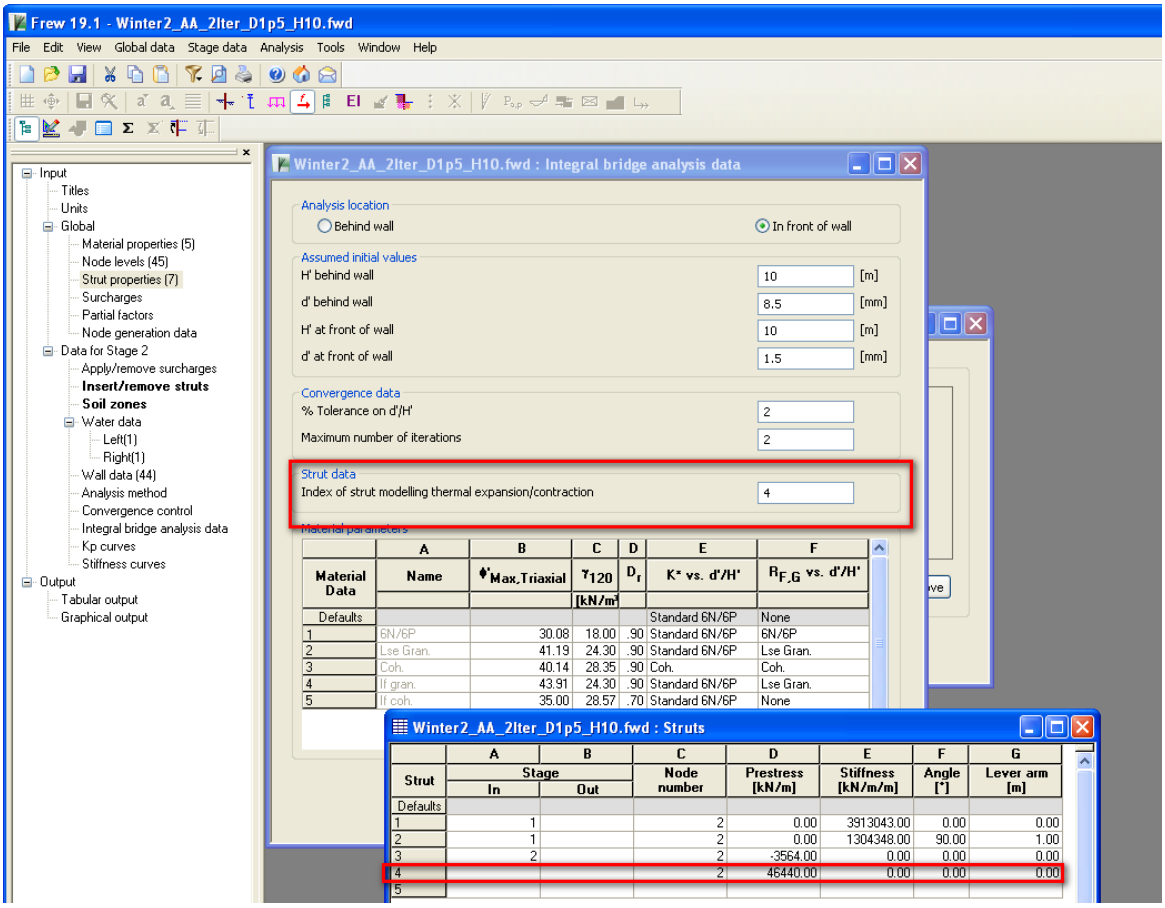
This would cause more items to be available in the Gateway. These new items are "Integral bridge data" "Kp vs D/H curves" and "Rf,g vs D/H curves". The user should enter the necessary data in these dialogs and tables.

© Oasys Ltd. 2014



4. The user has to specify whether the integral bridge analysis needs to be done at the front/back of wall. The user needs to specify the strut index which models deck contraction and expansion. Preferably, this strut should have only prestress, and no stiffness. During the integral bridge analysis cycles of contraction and expansion, it will apply the prestress in this strut with appropriate sign.

© Oasys Ltd. 2014


Frew 19.1 - Winter2_AA_2Iter_D1p5_H10.fwd
File Edit View Global data Stage data Analysis Tools Window Help
Input
Titles
Units
Global
Material properties (5)
Node levels (45)
Strut properties (7)
Surcharges
Partial factors
Node generation data
Data for Stage 2
Apply/remove surcharges
Insert/remove struts
Soil zones
Water data
Left(1)
Right(1)
Wall data (44)
Analysis method
Convergence control
Integral bridge analysis data
Kp curves
Stiffness curves
Output
Tabular output
Graphical output
Winter2_AA_2Iter_D1p5_H10.fwd : Integral bridge analysis data
Analysis location
Behind wall
In front of wall
Assumed initial values
H' behind wall 10 [m]
d' behind wall 8.5 [mm]
H' at front of wall 10 [m]
d' at front of wall 1.5 [mm]
Convergence data
% Tolerance on d'/H' 2
Maximum number of iterations 2
Strut data
Index of strut modelling thermal expansion/contraction 4
Material parameters
Material Data | Name | Φ'Max,Triaxial | γ120 [kN/m³] | Dr | K* vs. d'/H' | RF,G vs. d'/H'
Defaults | Standard 6N/6P | None
1 | 6N/6P | 30.08 | 18.00 | .90 | Standard 6N/6P | 6N/6P
2 | Lse Gran. | 41.19 | 24.30 | .90 | Standard 6N/6P | Lse Gran.
3 | Coh. | 40.14 | 28.35 | .90 | Coh. | Coh.
4 | If gran. | 43.91 | 24.30 | .90 | Standard 6N/6P | Lse Gran.
5 | If coh. | 35.00 | 28.57 | .70 | Standard 6N/6P | None
Winter2_AA_2Iter_D1p5_H10.fwd : Struts
Strut | Stage In | Stage Out | Node number | Prestress [kN/m] | Stiffness [kN/m/m] | Angle [°] | Lever arm [m]
Defaults
1 | 1 | | 2 | 0.00 | 3913043.00 | 0.00 | 0.00
2 | 1 | | 2 | 0.00 | 1304348.00 | 90.00 | 1.00
3 | 2 | | 2 | -3564.00 | 0.00 | 0.00 | 0.00
4 | | | 2 | 46440.00 | 0.00 | 0.00 | 0.00
5

5. Once the data has been entered, the user can go for the analysis in the regular way. When the user clicks the "Analyse" button, the program performs normal analysis for all the non-integral bridge stages. If these previous stages are analysed successfully, it will first apply initial winter contraction (about half the prestress specified for the deck strut), followed by cycles of full winter contraction, and full summer expansion until convergence in d/h values is achieved. First, d/h value is found for the left side. Subsequently, the above procedure is repeated on the right side. During the analysis, the program automatically adds the necessary stages.

6. After analysis, the user can view the actual material properties used in integral bridge stages in the results output as shown below:

© Oasys Ltd. 2014

Winter2_AA_2Iter_D1p5_H10.fwd : Output

Material parameters used in integral bridge analysis (Right side)

H(Right) = 14.0241 [m]
D(Right) = 0.5634 [mm]
D/H(Right) = 40.18E-6

Node	Level [m]	Material index	Density [kN/m³]	k_0	k_a	k_{ac}	k_p or k^*_d	k_{pc}	$R_{F,G}$	E [kPa]
16	*0.0*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*82424.8*
17	*-0.5000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*142764.*
18	*-1.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*184308.*
19	*-1.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*218076.*
20	*-2.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*247275.*
21	*-2.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*273372.*
22	*-3.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*297187.*
23	*-3.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*319230.*
24	*-4.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*339846.*
25	*-4.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*359282.*
26	*-5.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*377718.*
27	*-5.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*395296.*
28	*-6.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*412124.*
29	*-6.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*428292.*
30	*-7.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*443871.*
31	*-7.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*458922.*
32	*-8.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*473495.*
33	*-8.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*487632.*
34	*-9.000*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*501371.*
35	*-9.500*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*514743.*
36	*-10.00*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*527777.*
37	*-10.50*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*540496.*
38	*-11.00*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*552923.*
39	*-11.50*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*565076.*
40	*-12.00*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*576974.*
41	*-12.50*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*588631.*
42	*-13.00*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*600062.*
43	*-13.50*	*2*	*24.30*	*0.3414*	*0.2059*	*0.9075*	*0.4690*	*1.370*	*52.04*	*611279.*
44	-14.00	2	24.30	0.3414	0.2100	0.0	0.6300	0.0	N.A.	48045.0
45	-15.00	2	24.30	0.3414	0.2100	0.0	0.6300	0.0	N.A.	48045.0

Note: Italicized data correpsond to nodes in soil zone influenced by the abutment movement.

Stresses used/calculated in integral bridge analysis (Left side)

Node	Level [m]	Material index	Density [kN/m³]	k0	Ve [kPa]	Pe(At rest) [kPa]	Pe [kPa]
1	7.500	1	18.00	0.4988	2.250	1.122	1.418
2	7.000	1	18.00	0.4988	9.000	4.489	5.671
3	6.500	1	18.00	0.4988	18.00	8.978	11.35
4	6.000	1	18.00	0.4988	27.00	13.47	17.04

These results are printed for each stage below the deflection, bending moment, shear forces results.

Notes on Data Entry

When entering material data the user can choose to enter a granular or cohesive material.

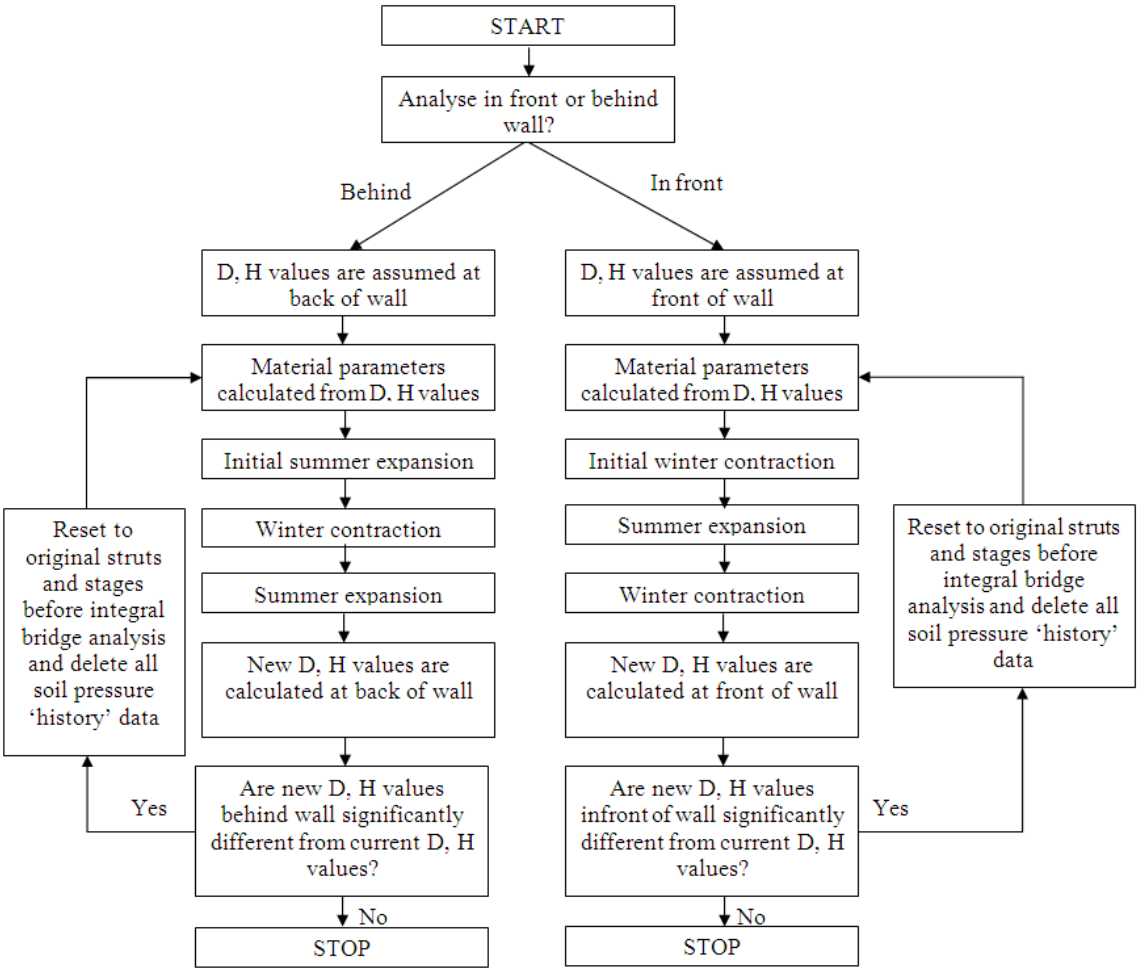
Where the granular option is chosen the material parameters are calculated as described in Appendix A of PD6694-1. The user is required to enter suitable stiffness curves (Rf,g vs D/H) and passive pressure (Kp vs D/H) curves. The Seed and Idriss (1970) curve for small strain stiffness of granular soils with 90% densification as shown in Appendix A of PD6694-1 is included as a standard curve for the calculation of small strain stiffness 'S&I 90% Densification' and this may be used if appropriate. Likewise there is an inbuilt option for Kp v D/H 'Standard 6N/6P' that calculates the Kp* value using the formula shown in section 9.4.3 of PD6694-1.

Where the cohesive option is chosen the material parameters specified in the materials table are used directly in the integral bridge analysis.

© Oasys Ltd. 2014

5.2 Algorithms

The following algorithm outlines the steps involved in integral bridge analysis in Frew:


START
Analyse in front or behind wall?
Behind
In front
D, H values are assumed at back of wall
D, H values are assumed at front of wall
Material parameters calculated from D. H values
Material parameters calculated from D. H values
Initial summer expansion
Initial winter contraction
Winter contraction
Summer expansion
Summer expansion
Winter contraction
New D, H values are calculated at back of wall
New D, H values are calculated at front of wall
Reset to original struts and stages before integral bridge analysis and delete all soil pressure 'history' data
Reset to original struts and stages before integral bridge analysis and delete all soil pressure 'history' data
Yes
Are new D, H values behind wall significantly different from current D, H values?
Are new D, H values infront of wall significantly different from current D, H values?
Yes
No
No
STOP
STOP

6 Seismic Analysis

Frew can be used to perform seismic analysis of a retaining wall. This analysis is undertaken based on Wood's method and the Mononobe-Okabe method. These are pseudo-static methods that estimate the additional lateral dynamic soil load on the wall. In this analysis struts representing the dynamic soil and groundwater loads are applied to the wall.

There are a range of methods available to assess the impact of seismic events on retaining walls, and the methods used by **Frew** will not be suitable in all cases. As a result it is important to confirm with a seismic analysis expert that the methods used are suitable before analysis is undertaken. Additionally loads other than the dynamic soil and groundwater may be applied, if there are likely to be other loads applied to the wall (e.g. due to adjacent structures) consideration will need to be given as to how these are taken into account.

© Oasys Ltd. 2014

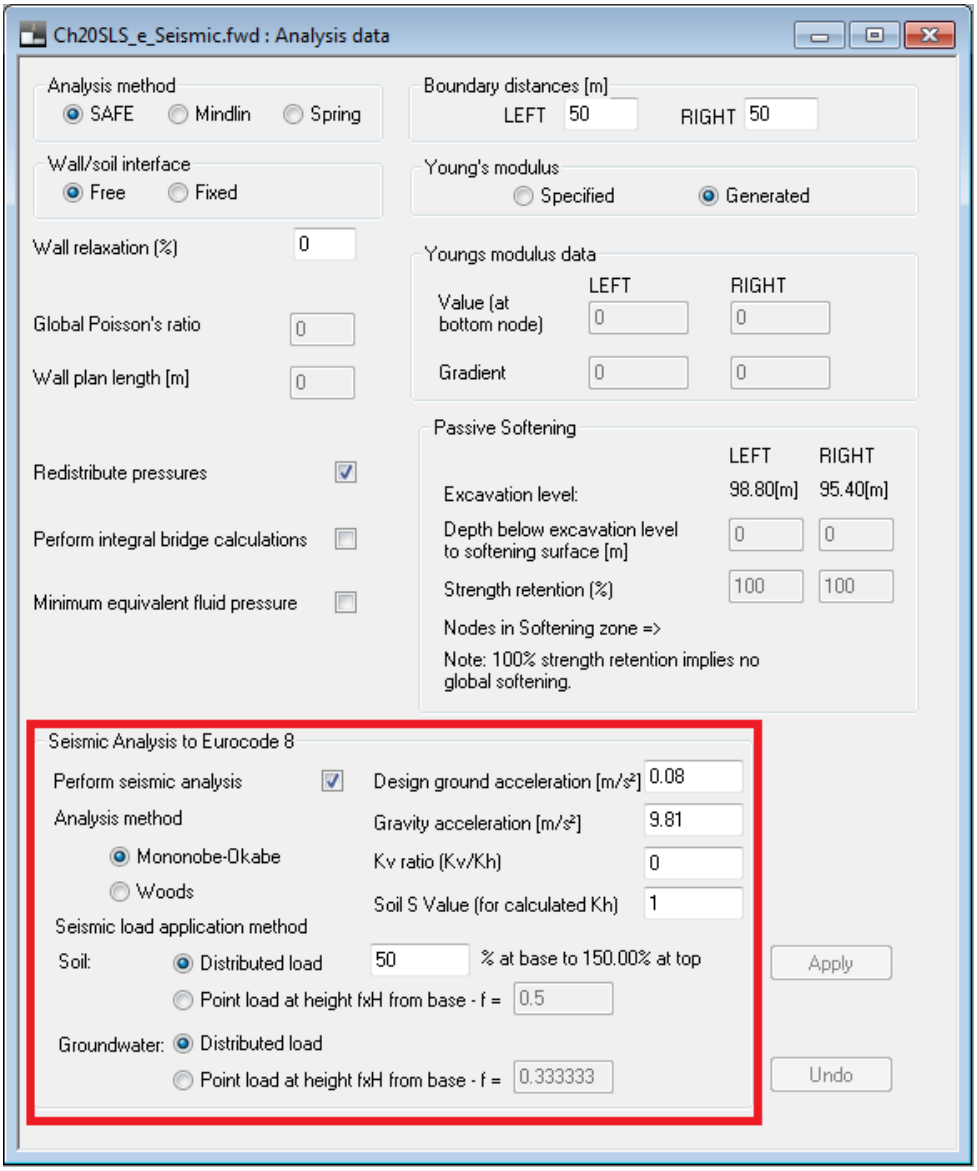
6.1 Data Entry

To use the seismic analysis feature the following steps must be followed.

1. First, set up a **Frew** analysis as normal.

2. Via the stage tree dialog add a further stage for the seismic analysis - this must be the final stage of the analysis.

3. Go to the 'Analysis method' dialog for the seismic analysis stage, and click on the 'Perform seismic analysis' check box.



4. Now select the preferred analysis method and method for load application. If you intend to use calculated Kh values it is also necessary to input the S value, and specify the design ground acceleration and acceleration due to gravity. Once the required data has been input click on 'Apply'.

© Oasys Ltd. 2014

Note: The analysis type and load application methods are described in more detail in the Seismic Analysis Methods section. Calculated Kh values are determined using the methodology described in Eurocode 8 (see Calculation of Seismic Coefficients), the S values for a range of stratigraphy types are given in Eurocode 8 Part 1.

5. Having chosen to perform seismic analysis, the 'Seismic material parameters' option becomes visible for the final stage in the gateway and stage tree dialog. Click on either of these to open the seismic material parameters table and then input relevant parameters as described below. Note that there is one entry in this table corresponding to each of the materials in the general material parameters table.

Parameter	Description
Description	This a non-editable field and gives the material description as per the main material parameters table.
Dry unit weight	Dry unit weight of the soil for calculation of lateral earth pressure.
Saturated unit weight	Saturated unit weight of the soil for calculation of lateral earth pressure.
Pervious/Impervious	Select either pervious or impervious. Pervious indicates that the soil is highly pervious to water flow and the load from the soil structure and water are calculated separately. Impervious indicates that water will move with the soil and that they will act together.
Derivation of Kh	Either user specified or calculated.
r	The r value for calculated Kh (see Table 7.1 of EN 1998-5:2004).
Kh	The lateral soil pressure coefficient. If user specified this must be entered manually, otherwise the calculated value will be shown in this box.
Fp	The Fp value is a dimensionless thrust factor used in Wood's method. This is frequently assumed to be 1.
Ess	Ess is the small strain stiffness of the soil. Where only small displacements of the retaining wall are anticipated an alternate small strain stiffness may be entered by the user.
Gradient Ess	The rate of change in small strain stiffness with depth. A positive value indicates stiffness increasing with depth. Note that the reference level for each material is as set in the general material parameters table.

6. Having set the seismic parameters next analyse the file. During analysis strut forces will automatically be generated and applied to the final stage representing the seismic force due to the soil movement. These strut forces can be reviewed in the Struts table following analysis, but will be deleted when results are deleted.

Note: To analyse intermediate construction stages it is necessary to create additional files for each stage that you wish to analyse. These files should be created as above, but with the final stage being that for which the seismic analysis is required, and with all subsequent stages deleted.

© Oasys Ltd. 2014

6.2 Seismic Analysis Methods

When analysing the forces generated by seismic ground movement, forces attributable the soil movement need to be considered, and potentially forces generated by the groundwater (where some or all of the retained soil is saturated, and it is highly pervious to movement of the groundwater). There are two principle methods in **Frew** to calculate the dynamic soil force, Mononobe-Okabe and Wood's method. The calculations undertaken by **Frew** to generate soil forces based on these methods are outlined in the sections Calculation of Seismic Coefficients, Wood's Method, Mononobe-Okabe Method and Load Application Methods. The calculations to determine the dynamic load from groundwater are covered in the section Groundwater Loading.

6.2.1 Calculation of Seismic Soil Pressure Coefficient

Horizontal Seismic Coefficient (Kh)

Where the calculated Kh option is chosen, Kh will be calculated using the following formula:

$$Kh = \alpha S/r$$

For which,

- α is the ratio of the design ground acceleration to acceleration due to gravity,
- S is the soil factor specified by the user,
- and r a factor representing the ratio between the acceleration value producing the maximum permanent displacement compatible with the existing constraints, and the value corresponding to the state of limit equilibrium.

Vertical Seismic Coefficient (Kv)

The vertical seismic coefficient is calculated using the ratio of Kv to Kh (Rk) and the horizontal seismic coefficient, such that:

$$Kv = Rk \times Kh$$

6.2.2 Wood's Method

Wood's method provides a simple calculation to determine the dynamic soil force (ΔP_d) on a retaining wall during a seismic event.

$$\Delta P_d = k_h \gamma H^2$$

For which k_h is the horizontal seismic coefficient, γ the soil unit weight and H the retained height of the soil.

Wood's method is supported by the research detailed in Wood (1973) which presents the results from a range of simulations calculating the maximum dynamic pressure on the back of rigid retaining walls. The method is then described in later works, e.g. Wood & Elms (1990).

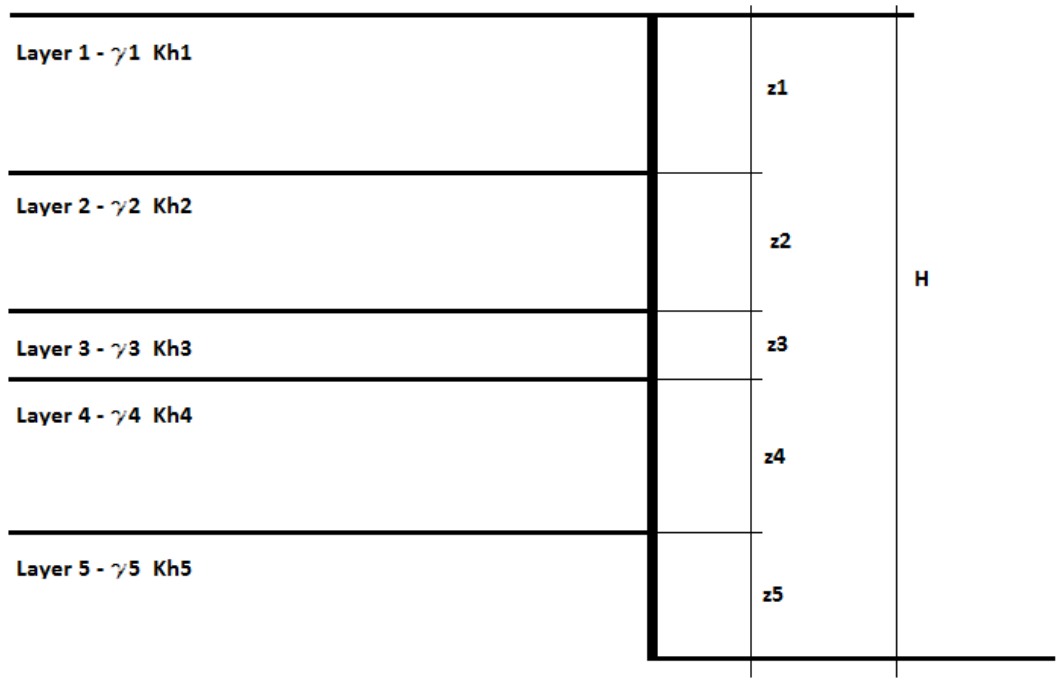
The simulations supporting this method assume a homogenous fully elastic retained soil and stiff

© Oasys Ltd. 2014

underlying material. As a result, where multiple strata are present behind the retaining wall a weighted average is taken for the parameters k_h and γ such that;

$$k_h = \frac{\sum z_n k_{hn}}{H} \quad \text{and} \quad \gamma = \frac{\sum z_n \gamma_n}{H}$$

Where n represents the layer number and z the layer thickness, as shown in the following example figure.



6.2.3 Mononobe-Okabe Method

The Mononobe-Okabe method is based on a classic coulomb wedge analysis, but with an enlarged active wedge taking account of the additional horizontal ground acceleration resulting in a change in the direction of the principle stress (θ). The method was first proposed in Okabe (1926) and Mononobe & Matsuo (1929) and has been developed since. The method detailed below and used in **Frew** is derived from BS EN 1998-5:2004 Annex E. Using this method the total force acting on the retaining wall (Ed) can be given by;

$$E_d = \frac{1}{2}\gamma(1 \mp k_v)KH^2 + E_{ws} + E_{wd}$$

For which K is the combined static and dynamic earth pressure coefficient, γ the soil unit weight, Kv the vertical seismic coefficient, H the retained soil height, Ews the static water force and Ewd the dynamic water force.

Given that the strut force should only represent the additional dynamic loading from the soil ΔPd and not the total load applied to the wall, the previous formula can be amended to give;

© Oasys Ltd. 2014

$$\Delta P_d = \frac{1}{2}\gamma(1 \pm k_v)(K - k_a)H^2$$

or for cases where Kv = 0,

$$\Delta P_d = \frac{1}{2}\gamma(K - k_a)H^2$$

The calculations in **Frew** are based on this, evaluating the additional force for each node along the face of the wall, then taking the sum of these to determine the total additional seismic force. This can be written as:

$$\Delta P_d = \sum_{x=1}^{x=n} \frac{1}{2}(\sigma'_{vxt} + \sigma'_{vxb})(1 \pm k_v)(K_x - k_{ax})z_x$$

For which x denotes each node, σ'_{vxt} is the effective stress at the mid-point of the element above, σ'_{vxb} is the effective stress at the mid-point of the element below, K_x is the combined static and dynamic earth pressure coefficient for relevant soil layer, k_{ax} is the active earth pressure for the soil at node x, and z_x is the distance from the mid-point of the element above to the mid-point of the element below.

To calculate the value of the combined pressure coefficient first the value of θ is calculated as shown below.

for unsaturated soils:

$$\theta = \tan^{-1}(k_h)$$

for saturated (impervious) soils:

$$\theta = \tan^{-1}\left(\frac{\gamma_s}{\gamma_s - \gamma_w}\frac{k_h}{(1 \pm k_v)}\right)$$

for saturated (pervious) soils:

$$\theta = \tan^{-1}\left(\frac{\gamma_d}{\gamma_s - \gamma_w}\frac{k_h}{(1 \pm k_v)}\right)$$

The combined earth pressure coefficient K can then be calculated based on the formulae shown below.

if $\beta \leq \Phi_d - \theta$

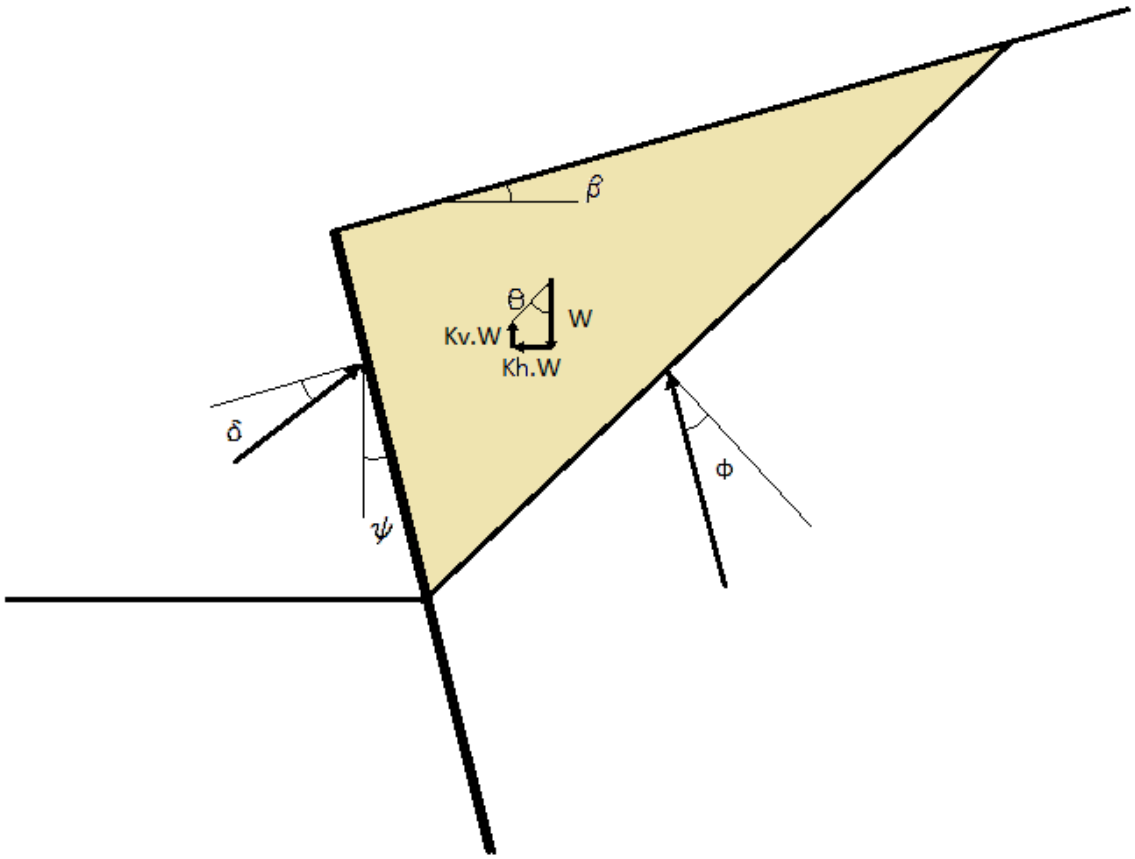
$$K = \frac{\sin^2(\psi + \phi_d - \theta)}{\cos\theta \sin^2\psi \sin(\psi - \theta - \delta_d)\left[1 + \sqrt{\frac{\sin(\phi_d + \delta_d)\sin(\phi_d - \beta - \theta)}{\sin(\psi - \theta - \delta_d)\sin(\psi + \beta)}}\right]^2}$$

or if $\beta > \Phi_d - \theta$

© Oasys Ltd. 2014

$$K = \frac{\sin^2(\psi + \phi_d - \theta)}{\cos\theta \sin^2\psi \sin(\psi - \theta - \delta_d)}$$

For which Φ_d is the design shear angle of the soil, δ_d is the angle of friction on the soil/wall interface, and the directions of the forces acting on the wedge and geometry of the wedge are as shown in the following diagram.



The active earth pressure coefficient is calculated as described in Calculation of Earth Pressure Coefficients.

In addition to any general limitations of the method, there are a number of points that the user should be aware of when using the Mononobe-Okabe method in **Frew**. These points should be considered and it should be confirmed that the assumptions made are valid for the model being assessed.

- **Frew** only considers the active case, where the soil stress along the face of the wall is at pressures greater than this then the loads generated will not be correct.
- Because **Frew** uses the vertical effective stress to calculate the dynamic soil force, the calculated force will be affected by any surface loads applied to the soil.
- **Frew** only considers vertical walls, i.e. it uses a value of 90° for ψ.
- Where partial factors are applied factored values of Φ and δ will be used.

© Oasys Ltd. 2014

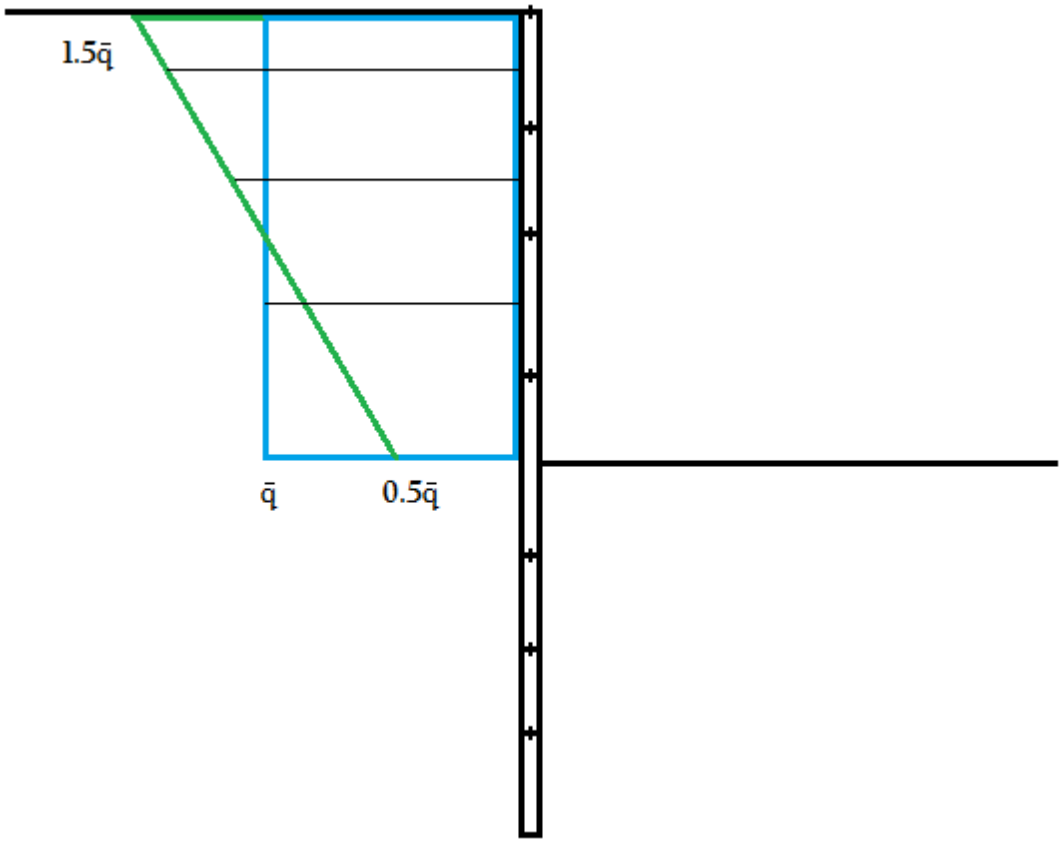
6.2.4 Load Application Methods

There are two options relating to how the seismic loading is applied to the retaining wall. The first is for the load to be distributed across the face of the wall.

Where the load is distributed it is applied as a strut load to each node along the retained soil (i.e. a strut is created with stiffness of 0 and a prestress equal to the required force and is applied to the relevant node). First the average pressure on the back of the retaining wall is calculated as;

$$\bar{q} = \frac{E_{wd}}{H}$$

The load at each node is then calculated. The load distribution is assumed to be linear and can be set by specifying the % of the average load at the base. **Frew** will then calculate the corresponding load at the top of the wall to ensure the the total load is unchanged. The force applied at each node is taken as the sum of the pressure as described above from the mid-point of the element below to the mid-point of the element above the node. Two example distributions are shown below, one with 100% average load at the base of the wall, i.e. constant pressure, and one with 50% of the average pressure at the base of the wall, i.e. 0.5q at the base to 1.5q at the top.



Where the load is applied as a point load, a strut force will be generated and applied at the elevation specified by the user. If nodes are generated automatically this will be taken into account when generating nodes to ensure that a node is present at the correct level.

© Oasys Ltd. 2014

6.2.5 Groundwater Loading

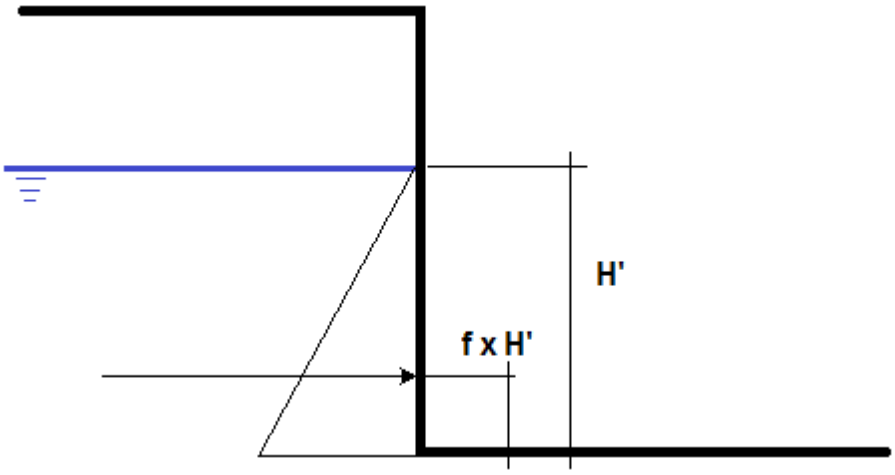
Where pervious soils are specified by the user the dynamic load from groundwater (Ewd) is calculated as:

$$E_{wd} = \frac{7}{12} k_h \gamma_w H'^2$$

For which Kh is the horizontal seismic coefficient, γw the unit weight of water and H' the height of the water table from the base of the wall.

If load application is specified as a point load, then the load is applied as a point load at a level of f x H' from the base of the wall, where the factor 'f' is specified by the user. If the load application is specified as a distributed load then it is applied as a pressure increasingly linearly from 0 at the water table to the maximum pressure (p_{max}) at the base.

$$p_{max} = \frac{7}{6} k_h \gamma_w H'$$



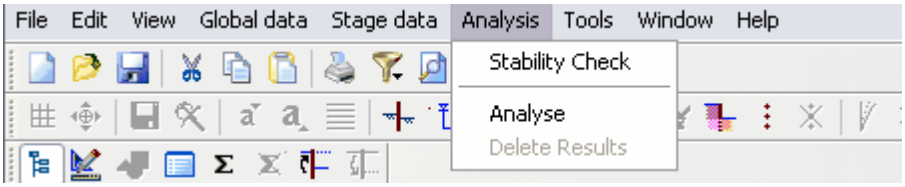
Where there are multiple strata present with some specified as pervious and others as impervious, the pressure profile is the same as that described above but with zero pressure applied along the length of the wall adjacent to impervious soils.

7 Output

7.1 Analysis and Data Checking

For a stability check, to check or set a wall toe level, select Stability Check from the Analysis menu, the button, or the Stability Check button on the Node Generation Data dialog.

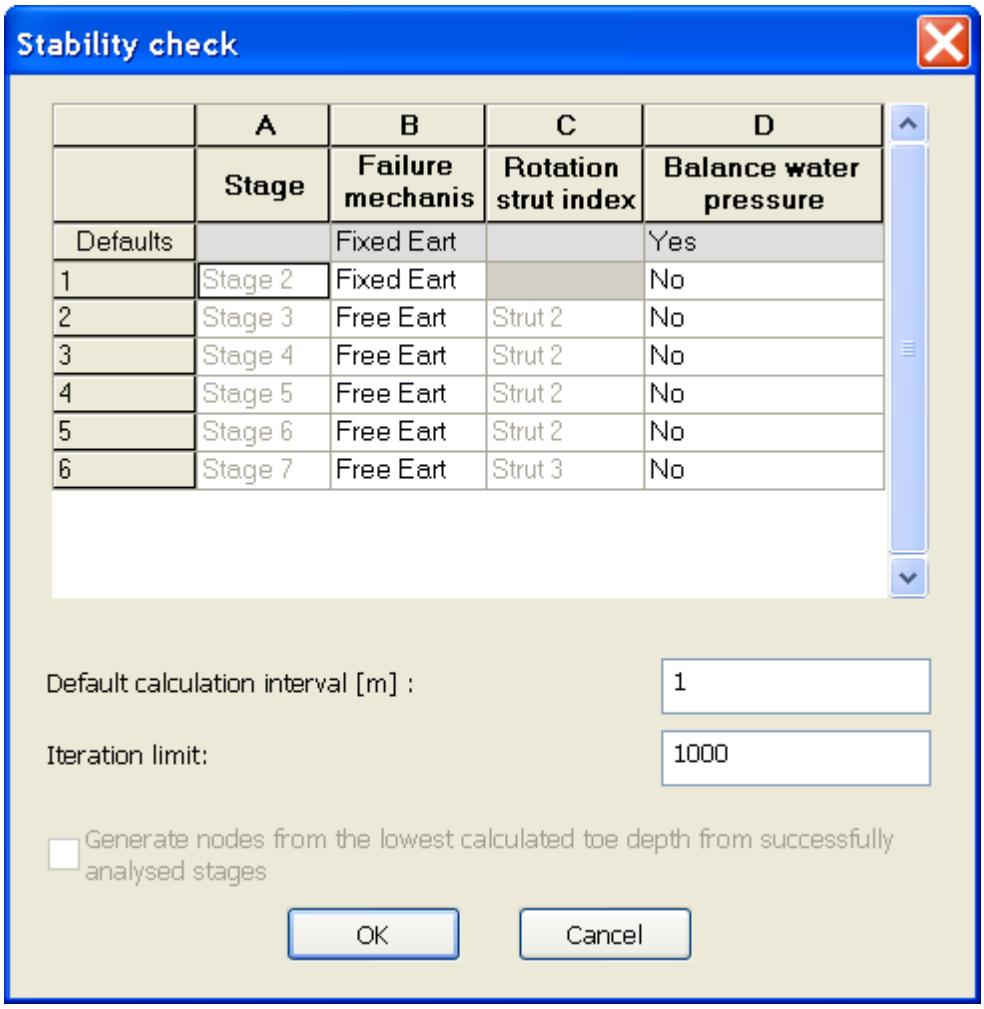
© Oasys Ltd. 2014



For full analysis, select Analyse from the Analysis menu or the Σ button.

Stability Check

A dialog will appear with default parameters for the stability check. The list of stages in the program, for which the stability check will be carried out will be listed in a table.



Select the required collapse mechanism (for details, see Stability Check). For the free earth method, the lowest strut is selected as the rotation strut. If automatic node generation is being used, ticking the "Generate nodes..." box will create all required nodes for the Frew analysis on successful completion of the stability check for at least a single stage.

The program initially uses the default calculation interval in the stability calculations. If the stability

© Oasys Ltd. 2014

calculations are not successful with the initial calculation interval, the program will appropriately change the value of the calculation interval, and re-run the stability check. This process is repeated at most three times.

Tip: If the stability check fails to find the toe depth within the specified number of iterations, try increasing the calculation interval or the iteration limit.

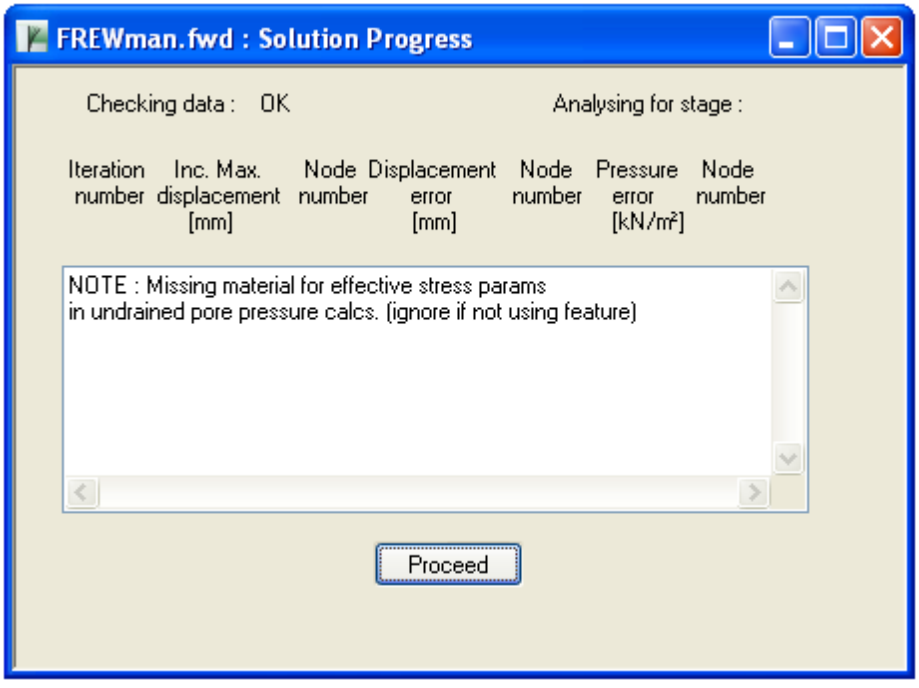
Full Analysis

The program carries out data checks as follows:

1. The wall is continuous i.e. does not contain any holes.
2. The soil is continuous beneath the surface.
3. The node spacing is reasonably uniform, in order to prevent any mathematical instability in the calculations.
4. All essential data has been entered, e.g. wall plan length and global Poisson's ratio for the Mindlin method; boundary distances for the **Safe** and Mindlin methods.
5. Nodes have been generated or specified.

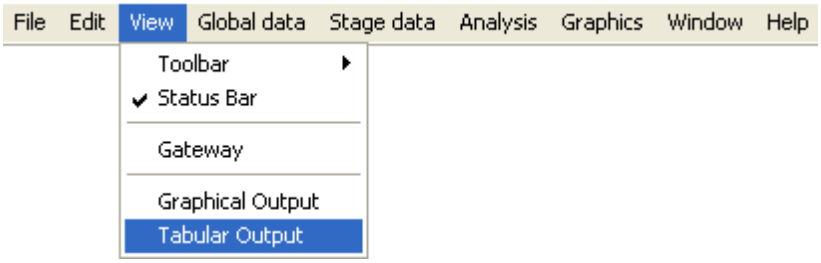
If there are errors, these must be corrected before the analysis can continue. Any data warnings will also be shown here. These should be reviewed and any required changes made. Sometimes these warnings will relate to features which are not required for the current analysis and can be ignored (for example, "NOTE: Missing material for effective stress params in undrained pore pressure calcs" is only relevant if undrained pore pressure calculations are required).

If no errors are found then the calculation continues through each stage. To continue to analysis when there are data warnings, click the "Proceed" button.



Note: The Tabular Output view will be shown once the calculations have been completed. It can also be accessed via View | Tabular Output as shown below, or the item in the Gateway.

© Oasys Ltd. 2014



7.2 Tabulated Output

Tabulated output is available from the View menu, the Gateway or the Frew toolbar. Two tabs are available for the tabular view, by default, the summary tab is shown that shows the key results and input data, however all data and available results are printed when selecting the full tab results output view. The items shown can be switched on or off by choosing Print Selection from the File menu or the filter button .

At the beginning of each stage's results, any surcharge or strut insertion or removal will be noted and the progress of convergence through the iterations is shown in a table. After the final stage's results, an additional table shows the "envelope" of the calculated displacement, bending moment and shear force values at each node.

Lines of output can be highlighted and then copied to the clipboard and pasted into most Windows applications (as shown below). The output can also be directly exported to various text or HTML formats by selecting Export from the File menu.

The results table is quite wide so the default font size is condensed. If larger size print is required, this can be set by clicking the Larger Font button on the toolbar. Note that the Page Setup may need to be landscape to avoid the lines of the results table scrolling on to two lines.

STAGE 9 : WALL & STRUT RELAXATION

RESULTS FOR STAGE 9 : Wall & Strut Relaxation

Surcharge or strut changes

```
Strut no. 3 removed at this stage
Strut no. 4 removed at this stage
Strut no 5 inserted at this stage
Strut no 6 inserted at this stage
```

Calculation details

```
E Profiles assumed for calculation (generated):
On the LEFT:  E at ground level = 38000.   E at bottom node = 53000. kN/m²
On the RIGHT: E at ground level = 39000.   E at bottom node = 58000. kN/m²
Iter   Inc Node   Disp Node   Press Node
 no.   max  no. error.  no.   error  no.
     displ
      [mm]        [mm]      [kN/m²]
   1   0.0    1 6.1375    1    0.14   11
```

© Oasys Ltd. 2014

2	6.1	1	0.1587	3	10.77	3
3	6.3	1	0.1620	4	7.65	4
4	6.4	1	0.1570	5	6.01	5
5	6.5	1	0.1503	5	5.00	6
10	6.9	1	0.0971	5	2.71	8
15	7.1	1	0.0484	6	1.32	9
20	7.2	1	0.0187	6	0.55	11
23	7.2	1	0.0092	6	0.26	11

Ground level left = 50.00 Ground level right = 40.50

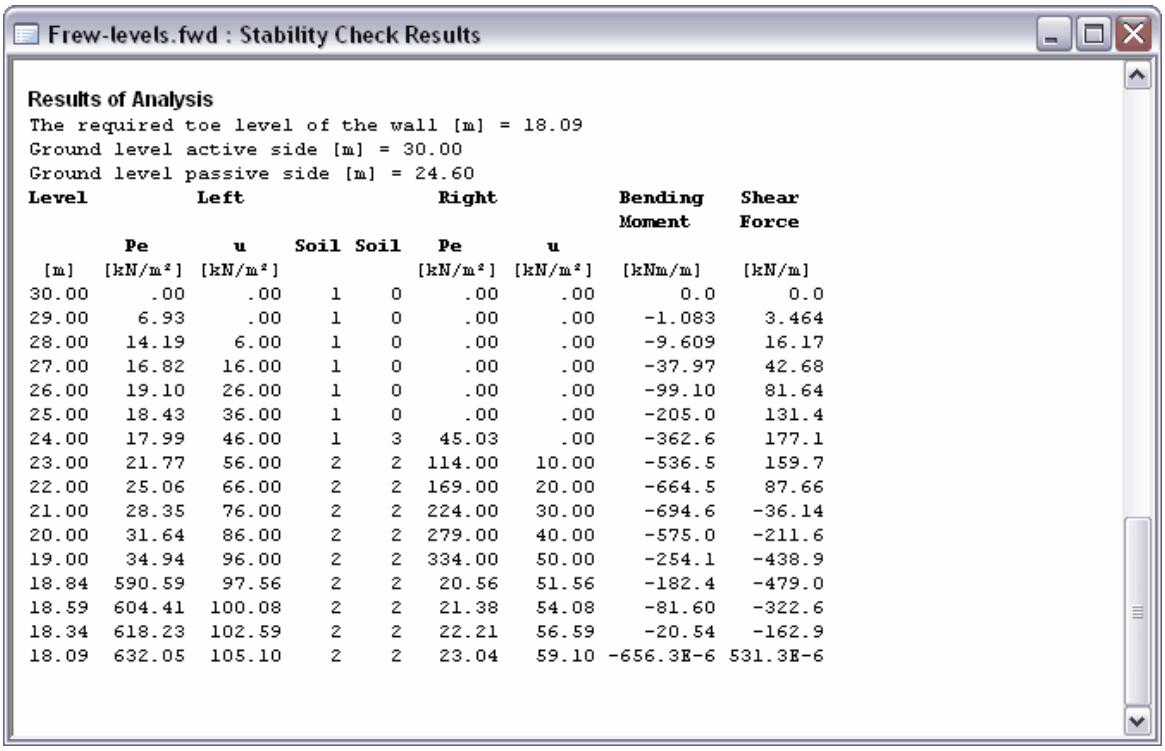
Node	Stress Level	Disp	Vt	Stress Pore Ve	Pt	Pore Pressure Pe	Pressure BM	Shear		Soil Left	Right	
	[m]	[mm]	[kN/m²]	[kN/m²]	[kN/m²]	[kN/m²]	[kN/m²]	[kNm/m]	[kN/m]			[kN/m²]
1	50.00	38.93	5.000	5.000	1.026	1.026	0.0			3	0	
	0.0	0.0	0.0	0.0	0.0	0.0	0.0					
	50.00											
						0.0	-144.5					
2	49.00	42.05	20.00	20.00	29.42	29.42	0.0			3	0	
	0.0	0.0	0.0	0.0	0.0	144.0	-129.3					
3	48.00	44.89	40.00	40.00	8.080	8.080	0.0			3	0	
	0.0	0.0	0.0	0.0	0.0	258.6	-110.6					
4	47.00	47.21	58.75	58.75	11.87	11.87	0.0		A	3	0	
	0.0	0.0	0.0	0.0	0.0	365.2	-99.97					
5	46.00	48.81	80.00	70.00	24.12	14.12	10.00		A	3	0	
	0.0	0.0	0.0	0.0	0.0	458.6	-81.35					
6	45.00	49.50	100.0	80.00	36.08	16.08	20.00		A	3	0	
	0.0	0.0	0.0	0.0	0.0	527.9	-51.25					
7	44.00	49.15	120.0	90.00	48.01	18.01	30.00		A	3	0	
	0.0	0.0	0.0	0.0	0.0	561.1	-9.205					
8	43.00	47.69	140.0	100.0	59.92	19.92	40.00		a	3	0	
	0.0	0.0	0.0	0.0	0.0	546.3	44.76					
9	42.00	45.16	160.0	110.0	71.85	21.85	50.00		a	3	0	
	0.0	0.0	0.0	0.0	0.0	471.6	110.6					
10	41.00	41.71	180.0	120.0	83.78	23.78	60.00		a	3	0	
	0.0	0.0	0.0	0.0	0.0	325.0	188.5					

... etc

© Oasys Ltd. 2014

7.2.1 Stability Check Results

The following results are available in the Stability Check results:



Frew-levels.fwd : Stability Check Results

Results of Analysis

The required toe level of the wall [m] = 18.09
Ground level active side [m] = 30.00
Ground level passive side [m] = 24.60

Level	Left				Right		Bending Moment	Shear Force
	Pe	u	Soil	Soil	Pe	u		
[m]	[kN/m²]	[kN/m²]			[kN/m²]	[kN/m²]	[kNm/m]	[kN/m]
30.00	.00	.00	1	0	.00	.00	0.0	0.0
29.00	6.93	.00	1	0	.00	.00	-1.083	3.464
28.00	14.19	6.00	1	0	.00	.00	-9.609	16.17
27.00	16.82	16.00	1	0	.00	.00	-37.97	42.68
26.00	19.10	26.00	1	0	.00	.00	-99.10	81.64
25.00	18.43	36.00	1	0	.00	.00	-205.0	131.4
24.00	17.99	46.00	1	3	45.03	.00	-362.6	177.1
23.00	21.77	56.00	2	2	114.00	10.00	-536.5	159.7
22.00	25.06	66.00	2	2	169.00	20.00	-664.5	87.66
21.00	28.35	76.00	2	2	224.00	30.00	-694.6	-36.14
20.00	31.64	86.00	2	2	279.00	40.00	-575.0	-211.6
19.00	34.94	96.00	2	2	334.00	50.00	-254.1	-438.9
18.84	590.59	97.56	2	2	20.56	51.56	-182.4	-479.0
18.59	604.41	100.08	2	2	21.38	54.08	-81.60	-322.6
18.34	618.23	102.59	2	2	22.21	56.59	-20.54	-162.9
18.09	632.05	105.10	2	2	23.04	59.10	-656.3E-6	531.3E-6

- The **level** of each calculation point. The separation of these points is specified in the Analysis Options.

For the **Front** and **Back** of the wall:

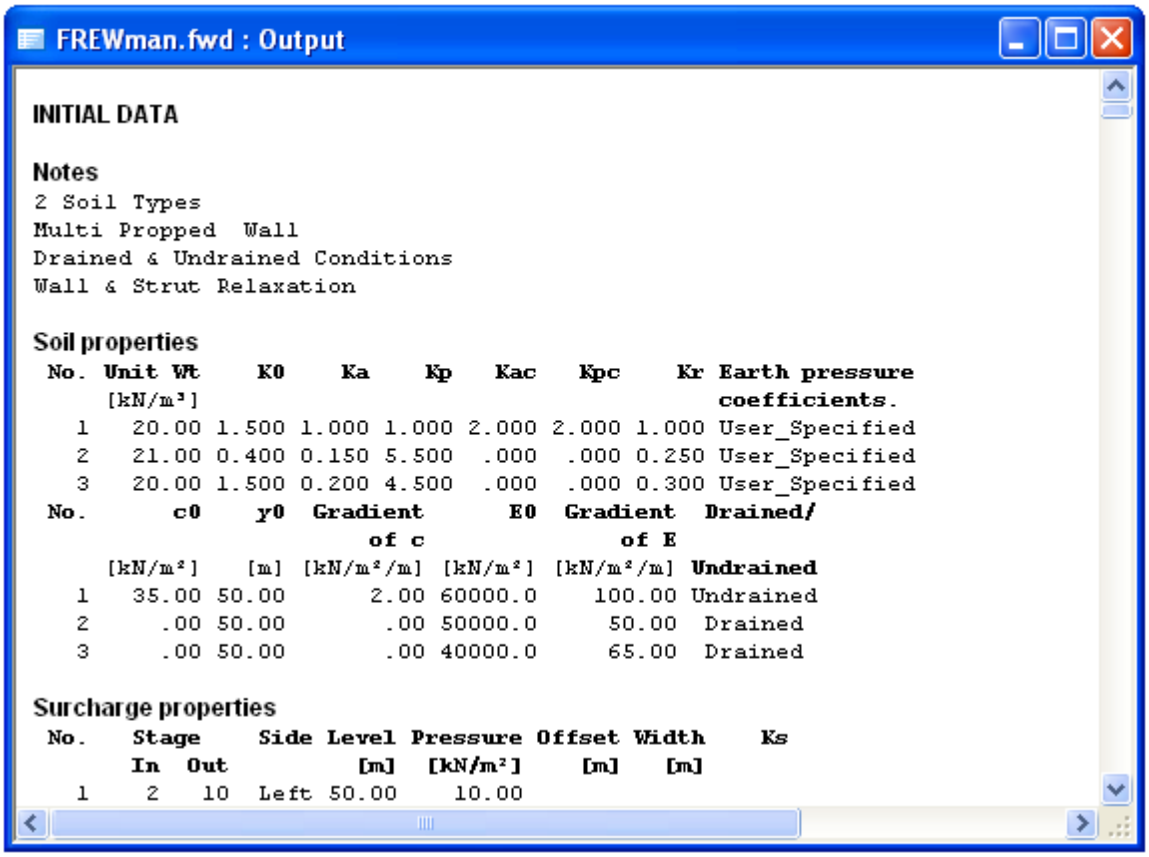
- Active and passive pressures (P_e),
- Pore water pressure (u) and material layer number.
- The **Bending moment** and **shear force** profiles down the wall.

If the "Balance water pressures" feature is switched on (see Analysis and Data Checking), two sets of results will be shown - the original results with the water data input by the user, and the final results obtained by the program after balancing the water pressure at the base of the wall.

© Oasys Ltd. 2014

7.2.2 Detailed Results

The output provides detailed results for all stages.



FREWman.fwd : Output

INITIAL DATA

Notes

2 Soil Types
Multi Propped Wall
Drained & Undrained Conditions
Wall & Strut Relaxation

Soil properties

No.	Unit Wt [kN/m³]	K0	Ka	Kp	Kac	Kpc	Kr	Earth pressure coefficients.
1	20.00	1.500	1.000	1.000	2.000	2.000	1.000	User_Specified
2	21.00	0.400	0.150	5.500	.000	.000	0.250	User_Specified
3	20.00	1.500	0.200	4.500	.000	.000	0.300	User_Specified

No.	c0 [kN/m²]	y0 [m]	Gradient of c [kN/m²/m]	E0 [kN/m²]	Gradient of E [kN/m²/m]	Drained/ Undrained
1	35.00	50.00	2.00	60000.0	100.00	Undrained
2	.00	50.00	.00	50000.0	50.00	Drained
3	.00	50.00	.00	40000.0	65.00	Drained

Surcharge properties

No.	Stage In	Stage Out	Side	Level [m]	Pressure [kN/m²]	Offset [m]	Width [m]	Ks
1	2	10	Left	50.00	10.00			

Listing of additions or removal of struts or surcharges for each stage

```
Strut no. 1 inserted at this stage
Surcharge no. 1 applied at this stage
```

Calculation details

E profiles used in the calculation
Progress through iterations, showing maximum incremental node displacement, displacement error and pressure error and where they occur

Ground levels front and back for each stage

Profile down all nodes of

- Displacement
- Vertical total and effective stress (V_t and V_e)
- Horizontal total and effective stress (P_t and P_e)
- Water Pressure (U)
- Bending Moment
- Shear Force

© Oasys Ltd. 2014

Note: For the undrained condition, if undrained pore pressures are not calculated by the program, the values of Ve and Pe shown and the user's U value will be apparent effective stresses and pore pressures rather than actual stresses and pore pressures. A note will be added to the foot of the results table and the values shown in brackets. See Undrained materials for more background.

7.2.2.1 Results Annotations and Error Messages

Indicator symbols will be added to the results table if the soil pressure limits are being exceeded. These are as follows:

Indicator	Meaning
A	The effective earth pressure is less than 1.01 times the active limit, but within the convergence pressure limit
P	The effective earth pressure is greater than 0.99 times the passive limit, but within the convergence pressure limit
*	The effective earth pressure is outside the convergence limits
a	The effective earth pressure is less than the Coulomb limit but still greater than the redistributed active pressure limit
p	The effective earth pressure is greater than the Coulomb limit but still less than the redistributed passive pressure limit
r	The effective earth pressure is greater than the redistributed passive pressure, but not sufficient to cause a failure to the surface (see Calculation of Active and Passive Limits and Application of Redistribution on page 58).
m	The earth pressure reported is the Minimum Equivalent Fluid Pressure (MEFP) where this has been specified.

Note that a, p and r are only used when redistribution is used to calculate active and passive pressure limits.

Warning or error messages will be shown in both the Solution Progress window and on the detailed results, if any stage has failed to analyse or has high bending moments below the base of the wall. Most are self explanatory but some additional detail is given below.

Analysis not converged within specified number of iterations

Try increasing the number of iterations in the Convergence Control dialog for the relevant stage.

This error may occur in Stage 0 where a surcharge is applied at ground level before the wall has been installed. This occurs where it creates a discontinuity in the lateral earth pressures at the top node. Where this occurs this can usually be worked around by applying the surcharge at a level slightly below the ground surface.

Vertical effective stress < 0

© Oasys Ltd. 2014

This means the data is such that the pore pressure exceeds vertical total stress (see Total and Effective Stress). This is usually caused by a data input error.

Active > passive at iteration j on the L (or R) side at node n

If the iteration number (j) is 1, this is probably due to soil properties (and surcharges of limited extent) being prescribed such that the active pressure exceeds the passive pressure at the node indicated. If j is equal to 2 or more, this message usually implies the solution is becoming numerically unstable.

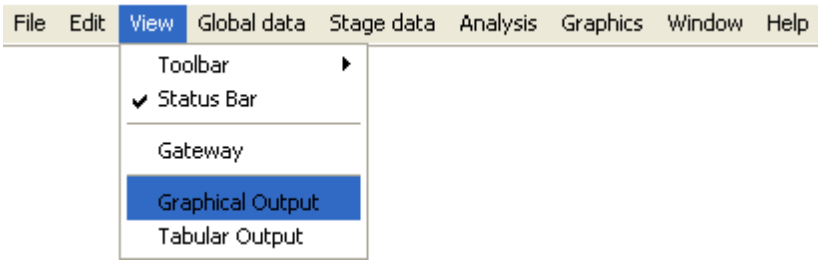
WARNING - Residual moment > 1% of peak moment in wall

WARNING - Wall base moment > 20% of average wall moment

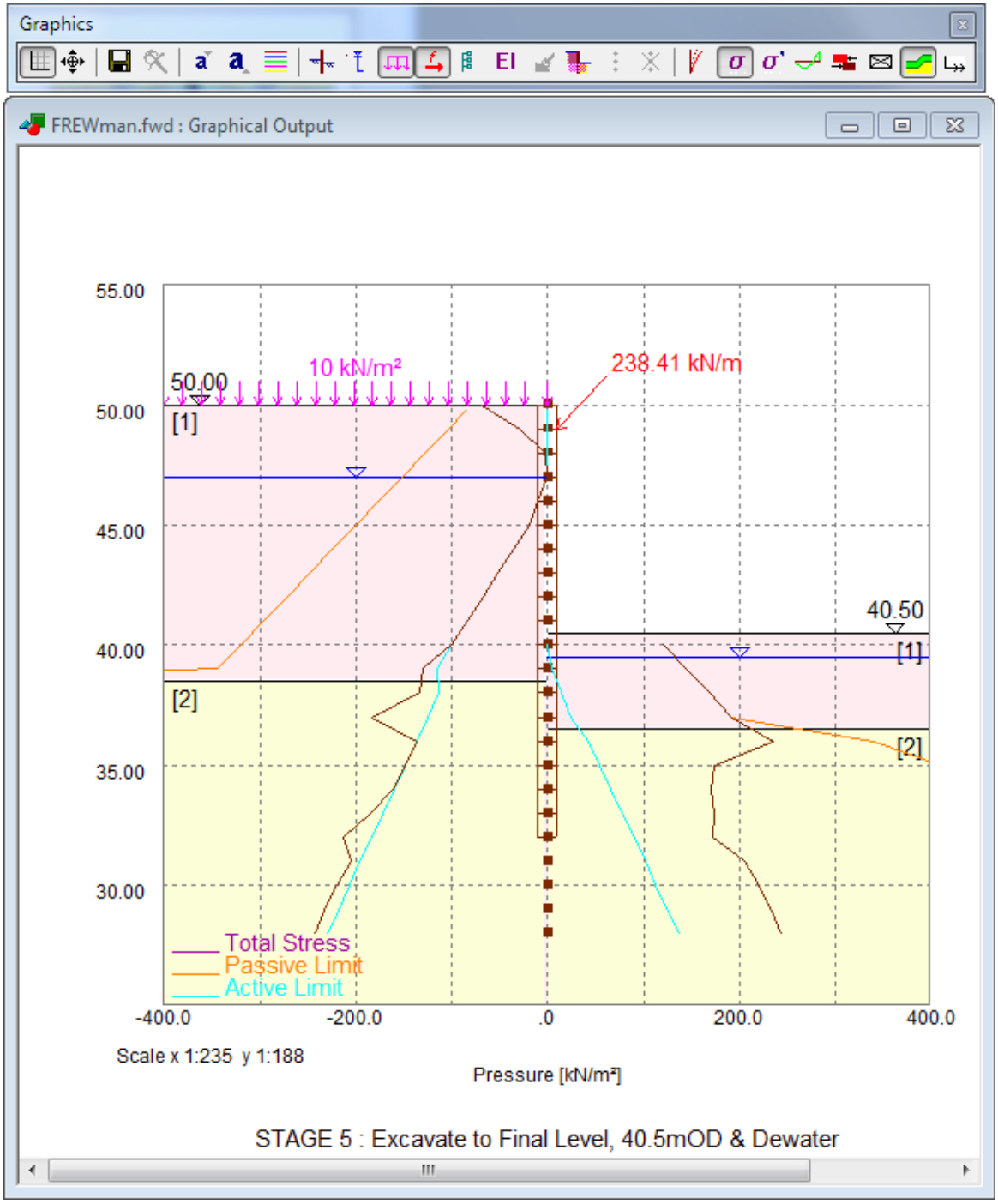
These warnings are output if the program obtains relatively high bending moments below the base of the wall. This can happen if the displacement of the wall is large compared with the flexure so curvature cannot be computed with sufficient accuracy (need to have several significant figures of difference of displacement and gradient of displacement between adjacent nodes.) The problem is generally caused by small stiff elements and can usually be overcome by increasing the distance between nodes. If these warnings are given, they indicate that **the wall is not in equilibrium and the results are not reliable.**

7.3 Graphical Output

Graphical output of the data and results is accessed via the View menu or the Gateway. The following provides details of the available graphics options.



© Oasys Ltd. 2014



Graphical toolbar buttons

Axis : Provides a reference grid behind the drawing.

Set Scale : This allows the user to toggle between the default 'best fit' scale, the closest available engineering scale. e.g. 1:200, 1:250, 1:500, 1:1000, 1:1250, 1:2500, or exact scaling. The same options are available via the View menu "Set exact scale" command.

© Oasys Ltd. 2014

Save Metafile :allows the file to be saved in the format of a Windows Metafile. This retains the viewed scale. The metafile can be imported into other programs such as word processors, spreadsheets and drawing packages.

Copy : allows the view to be copied to the clipboard in the form of a Windows Metafile.

Zoom Facility : Select an area to 'zoom in' to by using the mouse to click on a point on the drawing and then dragging the box outwards to select the area to be viewed. The program will automatically scale the new view. The original area can be restored by clicking on the 'restore zoom' icon as shown here.

Smaller/Larger font : allows adjustment of the font sizes on the graphical output view.

Edit colours: allows line and fill colours to be edited

Toggle strata on/off: switches strata fill colour on or off (for example, if printing to a monochrome printer you may prefer to switch the fill off)

Axes scaling : individual x-axis scales can be set for each plotted parameter. The same option is available via the View menu "Change axis scale(s)" command.

Deflection down wall.

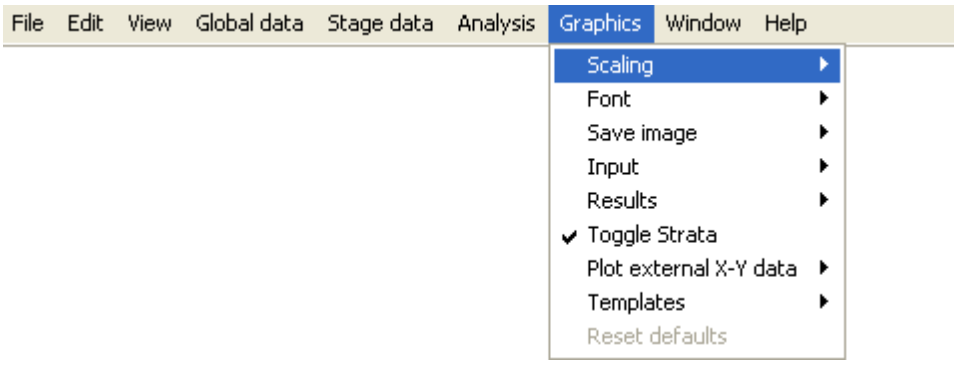
Active and passive total and effective stress profiles on either side of wall. The default is to show total stress. If effective stress is plotted, water pressure will also be shown.

Bending Moment profile down wall.

Shear Force profile down wall.

Envelope : Whereas other graphical results are for single stages, envelope provides the envelope of results for all stages in the calculation.

When the graphical output view is open the Graphics menu shows the following options.

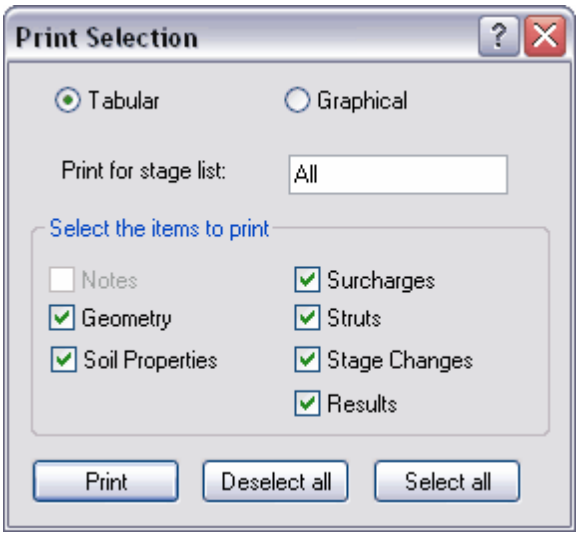


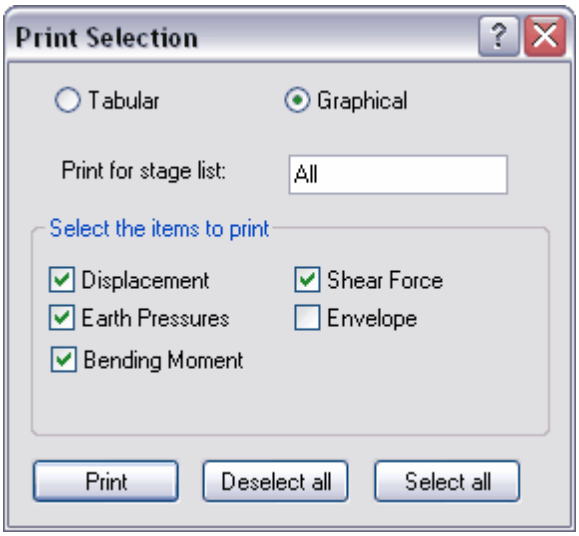
© Oasys Ltd. 2014

7.4 Batch Plotting

Tabular results or graphical plots for several stages at once can be batch printed using the filter button on the main toolbar.

This will show the Print Selection dialog. Choose Tabular or Graphical and the required data and/or results to show. Enter the required stage list. "All" is the default, but individual stages can be entered, separated by spaces, or ranges of stage e.g. "2 to 5".





For **tabular output**, the output view will be opened or updated with only the selected output shown. The information can be sent to printer in the usual way.

For **graphical output**, the Print dialog will be opened allowing selection of a suitable printer. **Note:** Graphical output can not currently be batch printed to PDF printer drivers - a warning message will be generated in this case. To print to PDF, open the graphical view and print each stage individually.

© Oasys Ltd. 2014

8 Detailed Processes in Frew

8.1 General

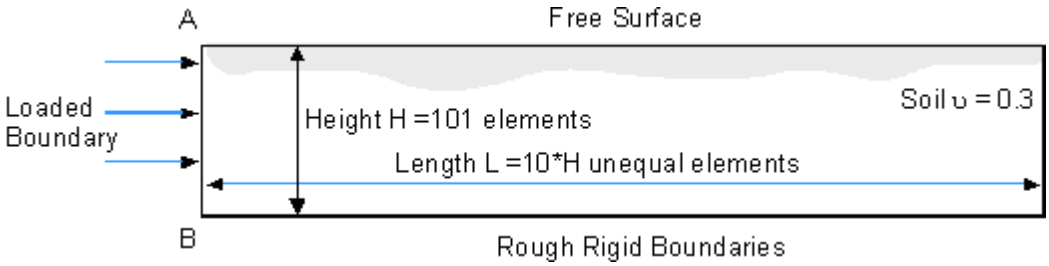
This section provides a detailed description of the assumptions and calculation methods carried out by **Frew**. It is important to gain a good understanding of these methods in order to be able to use and interpret the program to its full extent. Further discussion of the methods can be found in Pappin et al (1985).

8.2 Approximations Used in the Safe Method

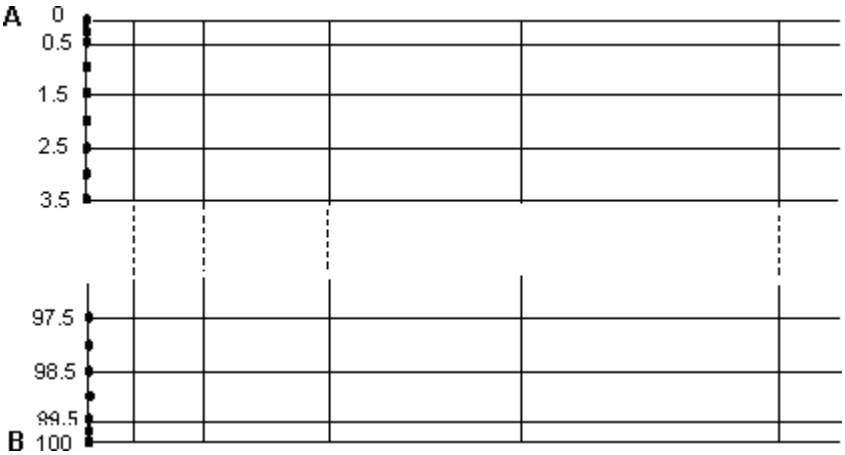
The **Safe** method uses a matrix of predetermined flexibility coefficients to represent the flexibility of the ground. The stiffness of the soil is then represented by inverting this flexibility matrix. The predetermined coefficients were generated using the finite element program **Safe**.

8.2.1 The Basic Safe Model

The flexibility coefficients stored in **Frew** were determined from a series of finite element analyses carried out using the **Safe** program.



The above figure shows the geometry and boundary conditions assumed for the mesh in the **Safe** analysis. The mesh is divided into 101 elements in height. The length is 10 times the total mesh height and is divided into a series of unequal elements, which increase in length away from the left hand boundary **AB**.



© Oasys Ltd. 2014

The model acts in plane strain and the vertical free face **AB** represents the location of the retaining wall in **Frew**.

The boundary AB is divided into 101 elements as shown. A unit force was applied to each element in turn, distributed as a uniform pressure over the length of the element.

The horizontal displacement at all nodes in the middle of the side of each element was then calculated, down the vertical free face **AB**. These displacements represent the flexibility coefficients and were stored as the flexibility matrix.

Using the principle of superposition, the total horizontal displacement at all nodes due to any load combination, can be estimated.

Two cases are considered - one with the nodes on the line AB free to move vertically and the other with the nodes fixed vertically. These are referred to as the "Fixed" and "Free" cases respectively. For each case there are two sets of flexibility coefficients stored within **Frew**. These apply to soils having either:

- a constant Young's modulus (E)
- or a Young's modulus which increases linearly with depth from zero at the surface.

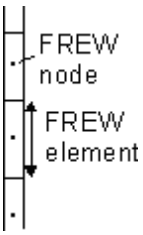
For varying profiles of E the user can specify their best estimate of a linear profile to best describe the variation with level. The program will combine and modify the matrices to accommodate this, see Accuracy with respect to Young's modulus (E).

Alternatively, the program will select a linear profile of Young's modulus, based on the specified non-linear profile; the program then applies further corrections as described in Irregular variation of E.

8.2.2 Application of the Model in Frew

Scaling factors are used to map the flexibility matrices from the **Safe** model onto the user defined model in **Frew**.

To carry out this 'mapping' the boundary AB is divided into "Frew elements".



Each Frew node, below the current ground level position, is assigned a Frew element. Boundaries occur mid-way between adjacent Frew nodes.

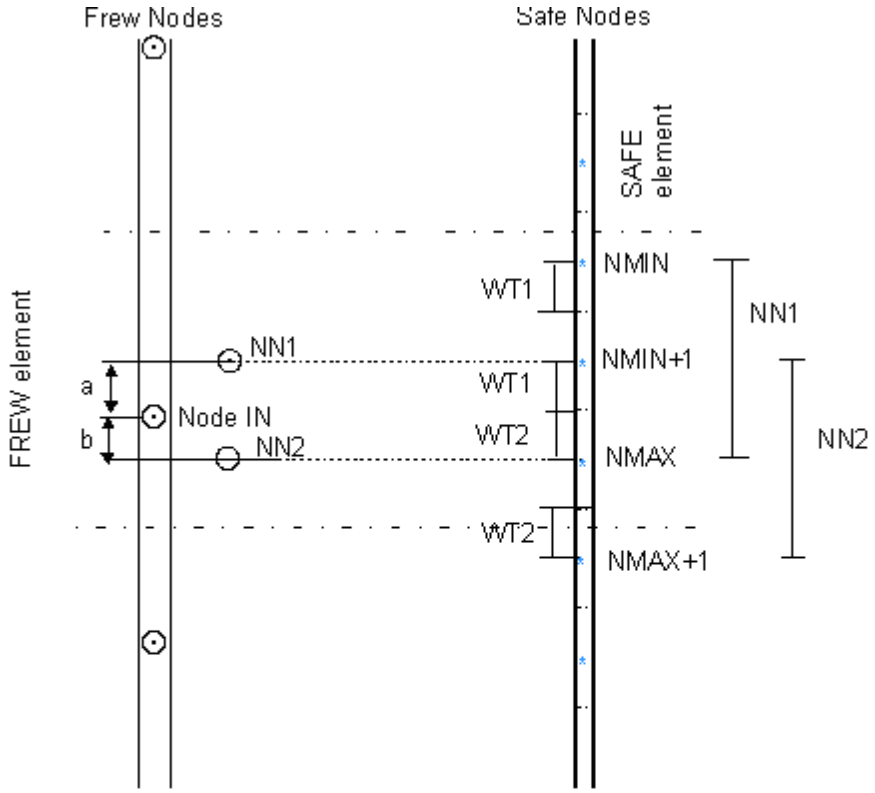
Note : Since the node spacing in **Frew** does not have to be uniform the node position is not necessarily at the mid-point of the Frew element.

The boundaries of the Frew elements are mapped onto the **Safe** model. The flexibility for a unit pressure over the Frew element length can then be determined.

This is achieved by summing the contributions, at the Frew node position, from all **Safe** elements

© Oasys Ltd. 2014

which contribute directly to the Frew element.



$$F(NN1,NN1) = F(NN1,NMIN) \times WT1 + F(NN1,NMIN+1) + F(NN1,NN1) + F(NN1,NMAX) \times WT2$$

$$F(NN2,NN2) = F(NN2,NMIN+1) \times WT1 + F(NN2,NMAX) + F(NN2,NN2) + F(NN2,NMAX+1) \times WT2$$

$$F(IN,IN) = \frac{[F(NN1,NN1) \times b + F(NN2,NN2) \times a]}{(NMAX - NMIN + 1 + WT1 + WT2)}$$

Note : Flexibility coefficient F(NN1,NMIN) represents displacement at node NN1 due to a unit load at node NMIN

Weighting factors are used to account for the effect for loading from a partial **Safe** element if Frew element boundaries bisect a **Safe** element. Using this procedure, the equivalent total load acting on the Frew node corresponds to the length of the Frew element, in multiples of the **Safe** element length.

Since the **Safe** coefficients were derived for 'elastic soil', the calculated Frew coefficients can be scaled by the ratio of the **Safe** element length to the Frew element length. This gives an equivalent total load at the Frew node of unity.

In general the Frew node will not correspond directly with a **Safe** node, and in such circumstances a second level of interpolation is implemented.

Two sets of Frew flexibility coefficients are calculated which correspond to the Frew element centred on the **Safe** nodes immediately above and below the Frew node. The actual Frew flexibility coefficients used in the calculations is then taken to be a weighted average of these two sets of coefficients.

© Oasys Ltd. 2014

8.2.3 Accuracy with Respect to Young's Modulus (E)

The flexibility matrices that have been computed using finite elements are effectively accurate for two situations:

1. Young's modulus constant with depth
2. Young's modulus increasing linearly from zero at the free surface.

No precise theory is available to enable accurate matrices to be derived for other cases, and various intuitive methods have therefore been adopted. These have been tested by comparing flexibility matrices computed by **Frew** with the results of additional finite element computations using **Safe**.

8.2.3.1 Linear Profile of E With Non-Zero Value at the Surface

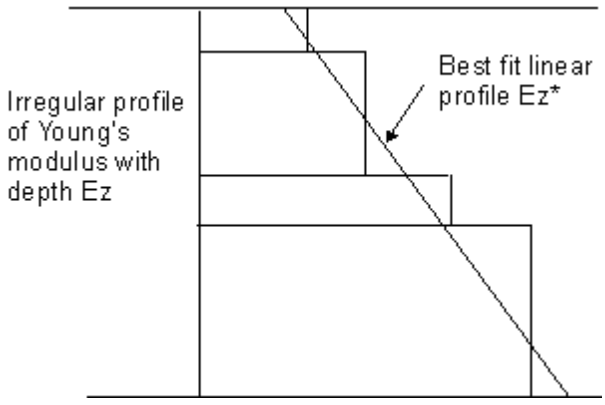
A case of particular importance is that of a linear profile of Young's modulus with a significant non-zero value at the free surface. It is found that very good results are obtained by dividing this profile into two components;

1. constant with depth
2. linearly increasing from zero.

The stiffness matrices for the two components are then added. No theoretical proof of this result has been found.

8.2.3.2 Irregular Variation of E

Linear variation of stiffness with depth can oversimplify the design profile. An approximate method of adjusting the matrices to accommodate irregular variations of soil stiffness has been determined empirically and consists of the following.



This method calculates a best fit linear Young's modulus profile E^*_z to represent the actual variation E_z.

© Oasys Ltd. 2014

Application of matrix

The flexibility matrix (F*), corresponding to the linear approximation, can then be derived from the pre-calculated matrices as described in The Basic Safe Model. In order to adjust this matrix to obtain the flexibility matrix (F) corresponding to the actual variation of Young's modulus each term in row i of (F*) is multiplied by a coefficient A_i. To maintain symmetry, terms F^*_{ij} and F^*_{ji} are both multiplied by the same coefficient, chosen as the smaller of A_i or A_j.

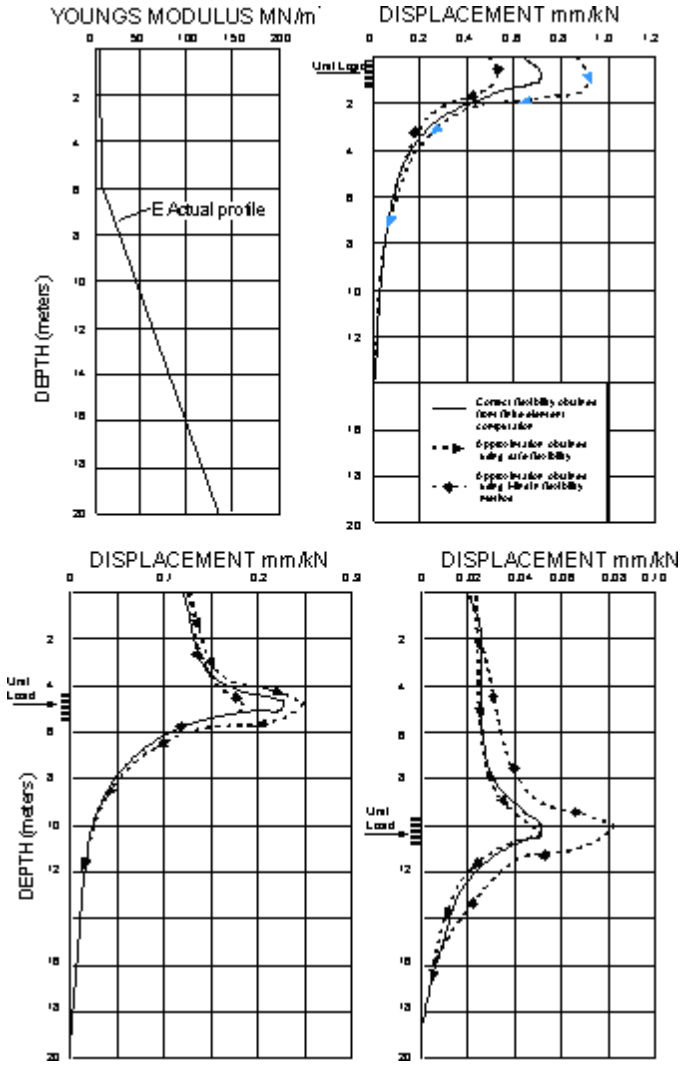
A number of alternative means of deriving coefficient A_i have been attempted based on consideration of the different distribution of work done due to unit load acting on two elastic soil blocks with Young's modulus profile E^*_z and E_z. The following expression has been developed for the coefficient A_i acting at node i.

$$A_i = \frac{\int_{z=0}^{D} \frac{E_z^{*2}}{E_z}\left(\frac{\partial \delta^*_{iz}}{\partial z}\right)dz}{\int_{z=0}^{D} E_z^*\left(\frac{\partial \delta^*_{iz}}{\partial z}\right)dz}$$

where δ^*_{ij} is the displacement at depth z of the elastic soil block with Young's modulus profile E^*_z due to unit load at node i.

No rigorous theoretical justification for this expression is available. However, comparison between finite element solutions and those produced by this approximation have been carried out and have shown that for most practical situations errors will rarely exceed 20%. The following figure shows one of the more severe cases that could be envisaged, (Pappin et al, 1985).

© Oasys Ltd. 2014



*Comparison of **Safe** and Mindlin flexibility approximations with FEA, at three depths*

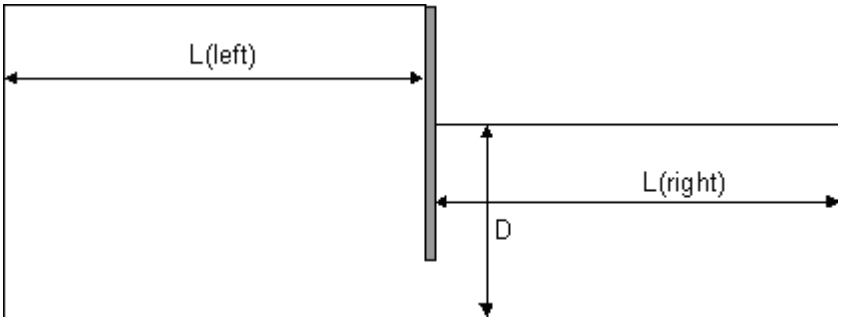
Here the displacement of the elastic soil block, with Young's modulus profile E_z, due to unit load at three different levels, is shown compared against rigorous finite element solutions.

8.2.4 Effect of the Distance to Vertical Rigid Boundaries

Vertical rigid boundaries may occur in the ground near a retaining wall due to unusual geological or man-made features. More often, the effect of a vertical boundary is required to model a plane of symmetry such as the centre line of the excavation.

The flexibility coefficients in the Safe model were derived for one specific geometric case which represented a ratio of L/D of 10, where L is the distance to the remote boundary and D the depth of soil in front of the wall.

© Oasys Ltd. 2014



Note : Clearly, as L/D changes the flexibility coefficients will change and hence the stiffness matrix. The greatest difference will occur at small ratios of L/D (i.e. large depth of soil in comparison to a close boundary). In this case it may be more reasonable to use a sub-grade reaction type of analysis where the spring length is well defined.

To allow for varying ratios of L/D the **Safe** method has been modified by adding a single spring at each node point. For high ratios of L/D the spring stiffness is small due to the large spring length. The results are then virtually identical to those of the elasticity method alone.

For small L/D ratios the single spring stiffness becomes dominant and controls and calculated wall movements.

8.2.4.1 Accuracy of Modelling Boundaries in Frew

Some test comparisons were carried out between **Frew** and **Safe**. These were to determine the error in **Frew** due to the simplified assumption of an elastic soil which has:

1. a constant **or**
2. linearly increasing stiffness with depth.

These comparisons are described below.

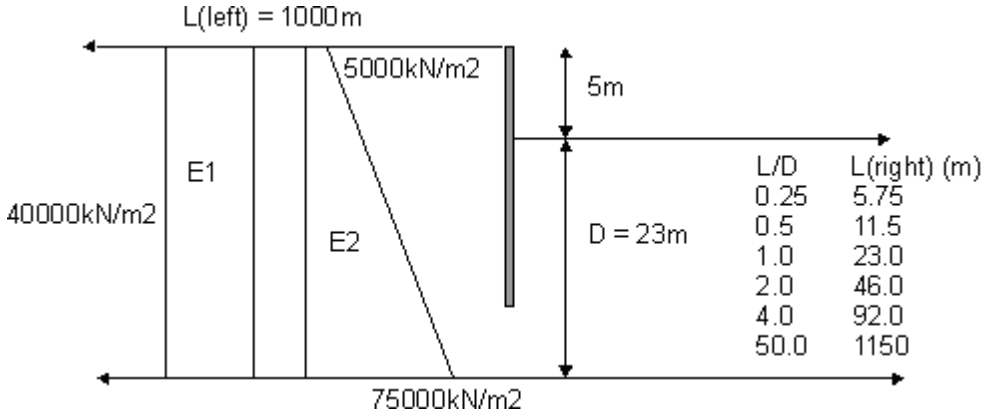
Changes in wall pressure computed by **Frew** and **Safe** were compared for the same wall movements for varying ratios of L/D. The comparisons were achieved by calculating the wall movements due to an excavation using **Frew**, and then using the calculated movements as input data for a finite element analysis using **Safe**. The behaviour was fully elastic in both cases.

With the same specified horizontal wall movements the **changes in stress** calculated by **Safe** are Δp_{Safe}. A comparison between Δp_{Frew} and Δp_{Safe} gives an indication of the agreement between the two methods of analysis.

The Model

In the **Frew** analysis the wall was taken as extending to a depth of 25m below ground level and the rigid boundary was at a depth of 28m. The dig depth was 5m giving 20m of soil in front of the wall. The distance to the remote boundary on the left side of the wall was taken as 1000m and on the right side the distance was varied to give L/D ratios of 0.25, 0.5, 1.0, 2.0, 4.0 and 50.0.

© Oasys Ltd. 2014



Comparisons were made for soil which has a constant Young's modulus (E1) throughout its depth equal to 40,000kN/m² and also for a soil in which Young's modulus (E2) increased linearly from 5,000kN/m² at ground level to 75,000kN/m² at a depth of 28m.

The soil is assumed to be dry and to have a unit weight of 20 kN/m³ and at rest coefficient of earth pressure $K_0 = 1.0$. For an excavation of 5m the change in horizontal stress is calculated as:

$$\Delta p = 5*20*K_r = 100K_r$$

therefore at some depth 'd' below the top of the wall the initial horizontal stress in the front of the wall will be:

$p_0 = 20d - 100K_r$ For d > 5m

For the test problem Poisson's ratio = 0.3 and

$$K_r = 0.3 / (1 - 0.3) = 0.43$$

therefore

$$p_0 = 20d - 43 \text{ kN/m}^3$$

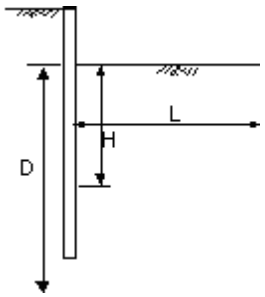
Due to digging 5m the wall moves and the stresses in front of the wall increases to p_f. The change in stress is therefore defined as

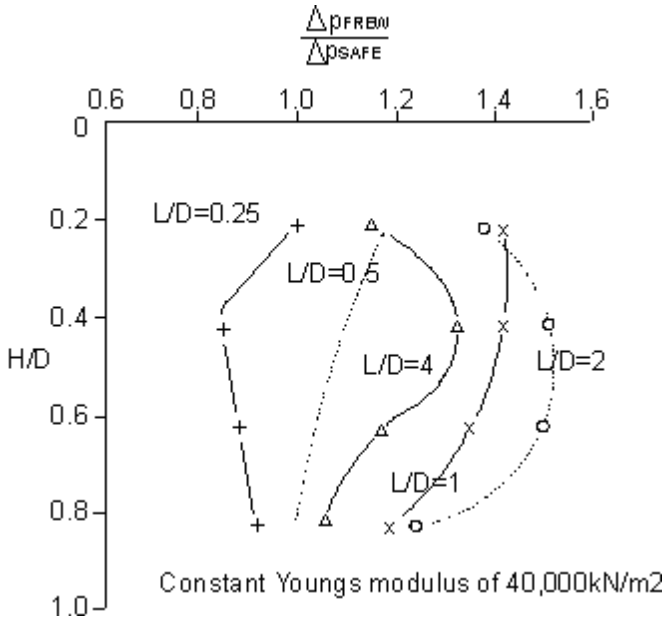
$$\Delta p_{Frew} = p_f - p_0$$

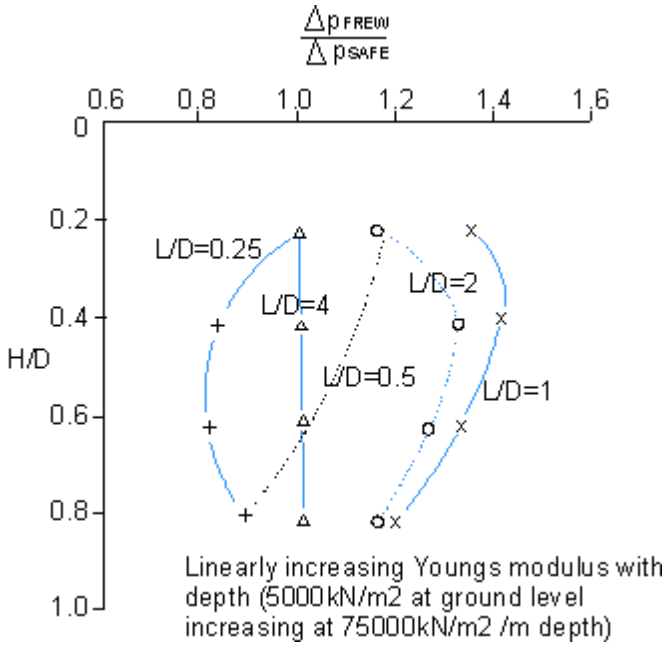
Summary of Results

The ratio of $\Delta p_{Frew} / \Delta p_{Safe}$ is shown below for the two cases considered.

© Oasys Ltd. 2014



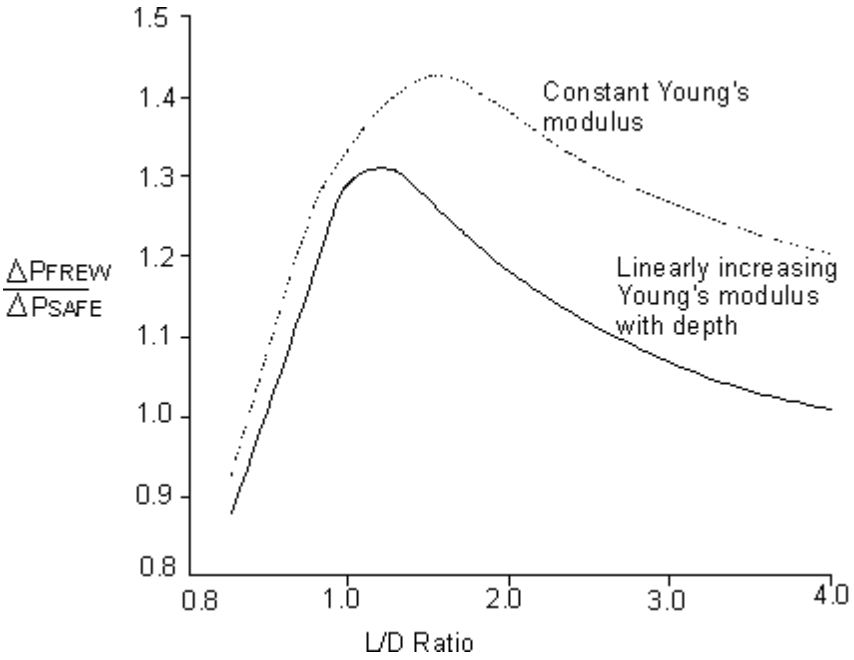




This shows how $\Delta p_{Frew} / \Delta p_{Safe}$ varies with depth and the ratio L/D. Taking an average ratio

© Oasys Ltd. 2014

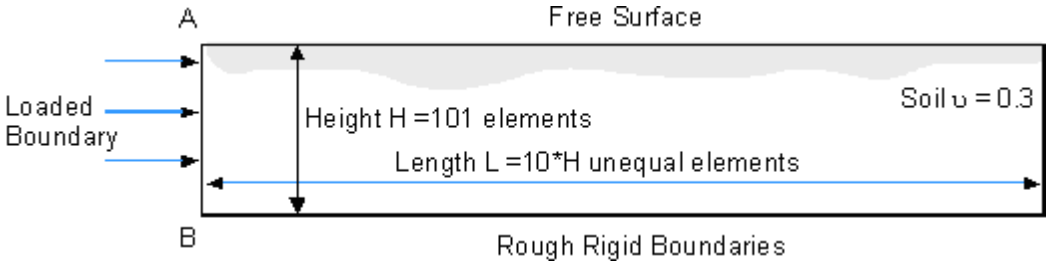
throughout the depth of soil the variation shown below is obtained.



8.2.5 Friction at the Soil/Wall Interface

Frew offers two options for representing the soil wall interface friction:

1. "**Free**" represents a frictionless interface
2. "**Fixed**" full friction.



***Safe** model*

These options select from the two sets of pre-stored flexibility matrices computed by **Safe** for the nodes on boundary AB. The two sets represent nodes **free** to move vertically or **fixed** vertically respectively.

© Oasys Ltd. 2014

8.2.5.1 Accuracy of the 'Fixed' Solution

In many situations when props or struts are being used, "fixed" and "free" give similar results.

An exception is a cantilever situation where the "fixed" method will give less displacements because it models greater fixity between the soil and wall.

It must be noted that the case with interface friction ("**fixed**") is somewhat approximate because Poisson's ratio effects are not well modelled. For example, these effects in a complete elastic solution, can **cause outward movement of the wall when there is a shallow soil excavation**.

8.3 Approximations Used in the Mindlin Method

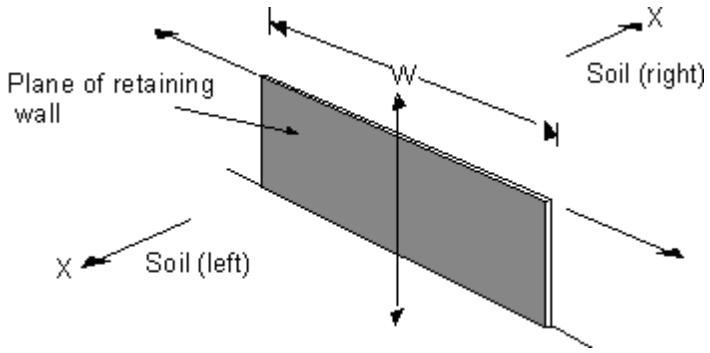
This method is similar to the "**Safe**" flexibility method in that the soil on each side of the wall is modelled as blocks of elastic material.

8.3.1 The Basic Mindlin Model

The method uses the integrals of the Mindlin equations which were published by Vaziri et al (1982). The integrals calculate the displacement, at any point, due to loading on either a vertical or horizontal rectangular area within an elastic half space.

If there were no rigid base or vertical loading the equations could be used directly to determine the flexibility coefficients of the nodal points due to horizontal pressures applied to the nodes, assuming that the wall is at a plane of symmetry.

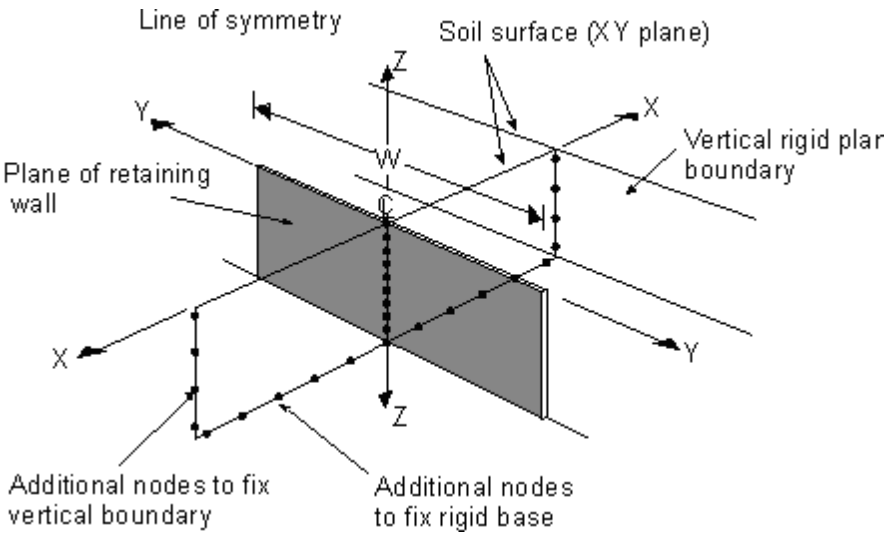
The flexibility of the soil, with each side of the wall taken separately, is equal to twice that of a half space. The effect of the width (W), or out of plane dimension of the retaining wall, can also be taken into account to some extent as the equations model the length of the pressure loaded rectangular area in the out of plane direction. Clearly if this dimension is large a plane strain condition is modelled.



© Oasys Ltd. 2014

8.3.2 Application of the Model in Frew

The soil being modelled in **Frew** by the Mindlin method, is not an elastic half space and the effects of the assumed rigid base and vertical boundary should ideally be incorporated. To take these boundaries into account additional boundary nodes are included when formulating the flexibility matrix, as below.



Half Space representation of a soil block

When modelling each side of the wall the soil must still be considered as a half space and the resulting flexibility matrix doubled.

Therefore to maintain symmetry at the plane of the wall additional nodes must be added to both sides. The base nodes are restrained both vertically (Z-Z direction) and horizontally (X-X) whereas the vertical boundary nodes are only restrained horizontally (X-X). As these nodes are on a plane of symmetry (X-X, Z-Z) they will not move in the (Y-Y) direction.

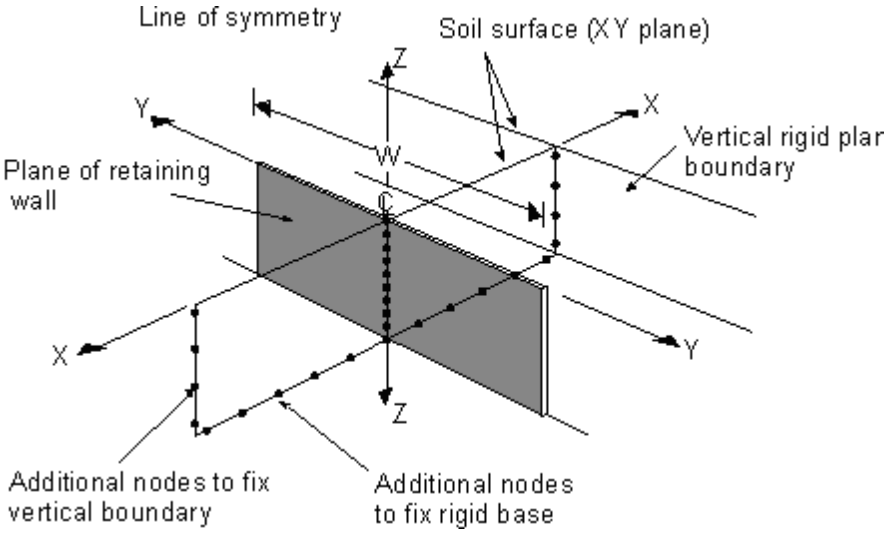
Nodal restraints are achieved by modelling stresses acting on rectangular areas centred at each boundary node to force the displacements of the boundary nodes to be zero. For a vertical boundary node a horizontal pressure is considered to act on a vertical rectangle. For a base node two stresses are considered, one being a horizontal traction and the other a vertical pressure, both acting on a horizontal rectangle. In all cases the width of the rectangle is taken as being equal to the width (W) specified for the wall.

The final soil stiffness matrix has been computed by eliminating the boundary nodes and inverting the flexibility matrix of the central nodes only.

8.3.2.1 Accuracy of the Mindlin Solution in Frew

This method of modelling fixity is considered to be reasonable when the width of the wall (W) is large relative to the depth or to the distance to the vertical boundary.

© Oasys Ltd. 2014

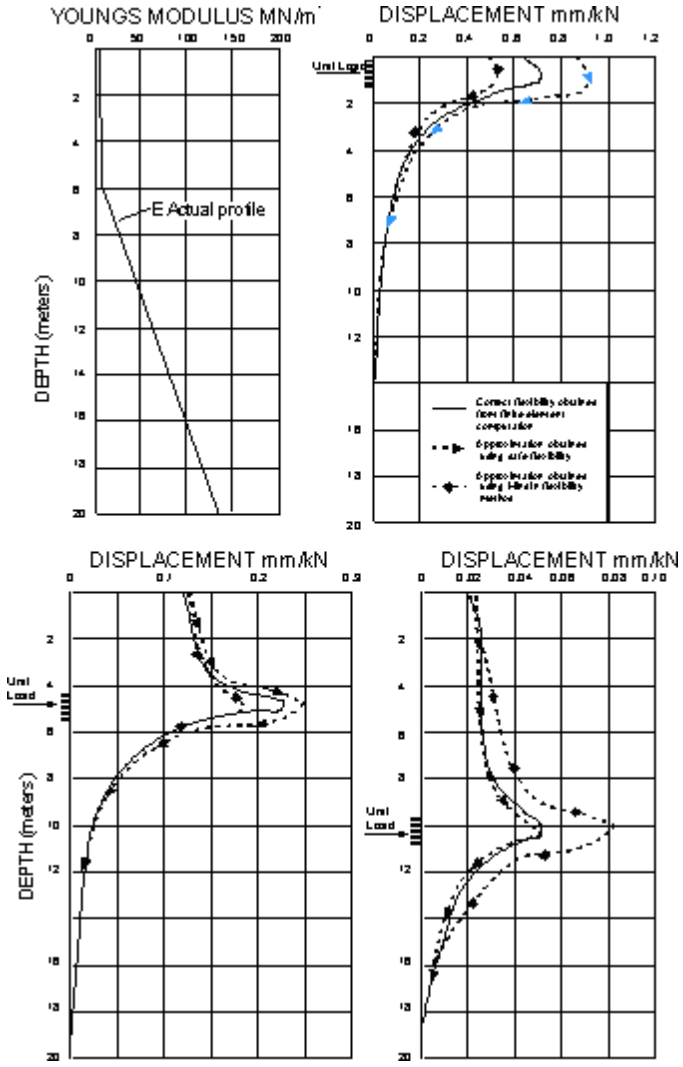


When W is small three dimensional effects will dominate and to approximate the fixity of a plane by a single line of nodes becomes somewhat dubious. Additional nodes on the fixed planes away from the plane of symmetry (X-X, Z-Z), or varying the width (W) of the loaded rectangle at the fixed nodes would improve this approximation. Nevertheless using the Mindlin flexibility method provides an approximate means of studying the importance of W.

A drawback of the Mindlin flexibility method is that **Young's modulus** is assumed to be constant with depth. This is significantly different from the "**Safe**" flexibility method which can model accurately a linearly increasing modulus with depth. Nevertheless the same ratios that are applied to model modulus variations can still be used with the Mindlin method, see Irregular variation of E.

An example comparing flexibility coefficients is given below, (Pappin et al, 1985).

© Oasys Ltd. 2014



*Comparison of **Safe** and Mindlin flexibility approximations with FEA, at three depths*

It can be seen that **Frew** provides quite good results when used with the Mindlin equations.

8.4 Calculation of Active and Passive Limits and Application of Redistribution

This section describes how active and passive limits are defined in **Frew**. The description refers to the requirements of certain parameters and equations being necessary and sufficient.

'necessary' denotes a condition which must be met in deriving a conservative solution.
'sufficient' denotes a condition which will ensure that a solution is conservative.

An item of information may therefore be necessary, but may also not be sufficient on its own to ensure a conservative solution. An ideal, accurate solution is both necessary and sufficient.

© Oasys Ltd. 2014

8.4.1 General

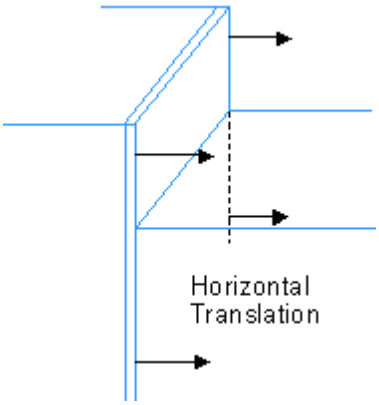
Approximations to the limiting pressures on a retaining wall may be calculated using either "lower bound" or "upper bound" methods.

- For the **lower bound** method, a set of equilibrium stresses which does not violate the strength of the soil, is studied. The limits which are calculated by this approach are **sufficient** for stability (ensuring a conservative solution), but may be unnecessarily severe.

Rankine used a lower bound method to calculate active and passive pressures in simple solutions. He assumed that wall **pressure** increased linearly with depth.

- In the **upper bound** method a failure mechanism is considered. The limits obtained are **necessary** for stability, but may not be sufficient.

Coulomb used an upper bound method to study the simplest failure mechanism - a plane slip surface - to derive the **forces** on a wall.



It is found that for the simplest case of all, a frictionless wall translated horizontally without rotation, their analyses give compatible results. This result is therefore accurate – both necessary and sufficient.

Note: For one slip surface Coulomb's method only yields the total force on the wall. In order to find the redistribution of that force, i.e. the pressures on the wall, further assumptions related to the mode of deformation are required.

For more complex problems, involving wall friction and complicated patterns of deformation, Rankine's simple assumptions about the pressure distributions are obviously wrong and Coulomb's planar slip surface is not the most critical. Many other researchers have therefore derived information about active/passive **forces** by studying other failure mechanisms. In the absence of additional assumptions, these methods yield the limiting force on the wall between the ground surface and any given point on the wall, but they do not dictate the distribution of that force, i.e. the **pressures** on the wall. They produce limits which are necessary, but not exactly sufficient. However, by seeking the most critical slip surface a result which is nearly sufficient (to ensure a conservative solution) is found.

© Oasys Ltd. 2014

8.4.2 Application in Frew

In **Frew**, **elasticity methods** are used to derive a pressure distribution on the wall, and this is then modified so that forces on sections of the wall are approximately within the limits required by plastic (strength) considerations.

Active limit for a dry cohesionless soil

The method used will first be described for dry, cohesionless soil, considering only the active limit.

For a uniform material, values of the coefficient of active earth pressure K_a have been derived by various researchers by searching for critical failure surfaces. These give necessary limits of the forces on the wall.

Strictly $$K_a = \frac{P_a}{\left(\frac{1}{2}\gamma' z^2\right)}$$

where :

P_a = the minimum effective soil **force** on the wall between the free surface and depth z

γ' = effective unit weight of soil.

Only if it is assumed that the earth pressure increases linearly with depth is it valid to use the same value of K_a in the equation for

wall pressure $= K_a \gamma' z$

Now define

$p_a = K_a \gamma' z$ in a uniform soil

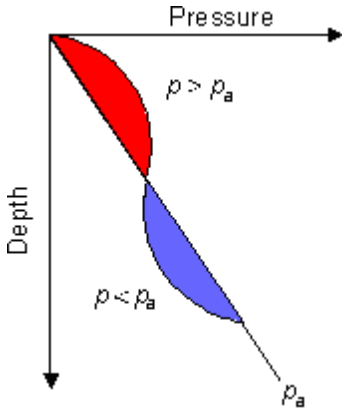
or

$$p_a = K_a \int_0^z \gamma' \, dz$$ in a non-uniform soil

Let earth pressure at depth z = p.

Then, provided the value of K_a is a very good upper bound the condition, $p \geq p_a$ at all depths, (E1) will be sufficient (safe), but not 'necessary' since the necessary condition only considers **force**.

© Oasys Ltd. 2014



Referring to the figure, the criterion of $p \geq p_a$ means that the pressure p cannot drop below the limit of p_a. This is the limiting condition used in **Frew** when "**no redistribution**" of the wall pressures is specified. It is sufficient to provide a conservative solution, but may be unnecessarily severe.

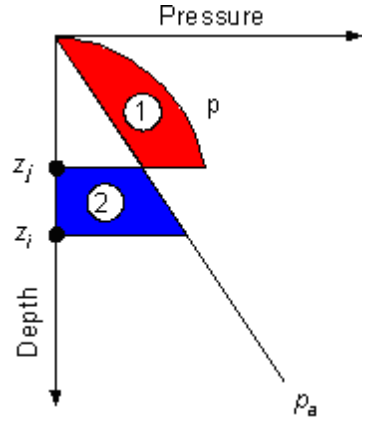
If *P* is the force on the wall between depth z and the ground surface, an alternative condition would be

$$\int_0^z p\,dz = P \geq P_a = 0.5K_a\gamma' z^2 \qquad \text{in a uniform soil} \qquad \text{(E2)}$$

$$\geq \int_0^z p_a\,dz = \int_{z'=0}^{z'=z} K_a \left[\int_0^{z'} \gamma'\,dz' \right] dz \qquad \text{in a non-uniform soil}$$

These equations would allow the type of stress distribution, indicated in the above figure, which would occur, for example, at a propped flexible wall.

The equation E1 for a uniform soil is necessary, but the following example shows that it is not sufficient.

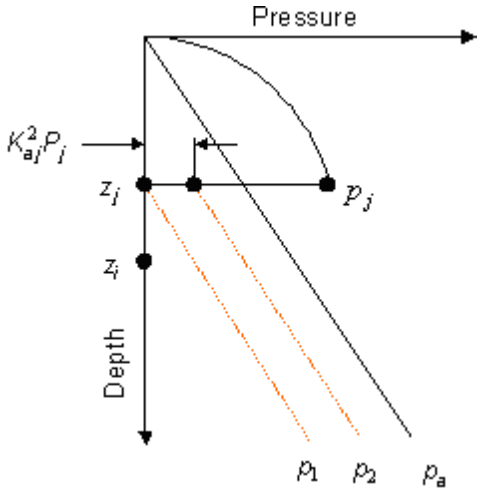


Consider the section of wall z_j z_i in the above figure. Above z_j the wall pressure p is in excess of P_a. Equation E2 would therefore allow the pressure between z_j and z_i to fall to zero, or even to have

© Oasys Ltd. 2014

negative values, provided that the area indicated as 2 does not exceed area 1.

This is clearly wrong; it is not admissible to have zero pressure on a finite length of wall supporting a cohesionless soil. If it were, the element of wall between z_j and z_i could be removed and no sand would flow out. It would be possible however, to have zero pressure at a point, with z_j and z_i coincident; it is admissible to have a very small hole in a wall supporting sand.



If z_j z_i is a finite length, sand would flow out because of the self-weight of the material between z_j and z_i. Thus, there is another limiting line, indicated as P_1 in the above figure. For depth z below z_j this limit is given by:

$$p \geq K_a \int_{z_j}^{z} \gamma' dz = p_1 \qquad \text{(E3)}$$

However, there is also a more severe restriction. This occurs because at depth z_j there must be a non-zero vertical stress since the horizontal stress immediately above z_j is $p_j \geq p_a$. In order to maintain this horizontal stress, the minimum vertical stress is approximately $K_a p_j$. Thus, for points below z_j, the line p_2 provides a limit:

$$p \geq K_a \left[\int_{z_j}^{z} \gamma' dz + K_{aj} p_j \right] = p_2 \qquad \text{(E4)}$$

Equations E2 and E3 are similar in form to $p \geq p_a$. It was argued in Application in Frew above that $p \geq p_a$ was sufficient but not necessary, whilst Equation E2 was necessary but not sufficient.

Similarly, Equation E4 is not necessary. By analogy with Equation E2, the necessary Equation becomes:

© Oasys Ltd. 2014

$$\int_{z_j}^{z_i} p\,dz \geq \int_{z_j}^{z_i} p_2\,dz = \int_{z_j}^{z_i} K_a \left[\int_{z_j}^{z} \gamma'\,dz + K_{aj}\,p_j \right] dz \qquad \text{(E5)}$$

When the redistribution option is specified in **Frew**, Equation E5 is enforced between all pairs of nodes corresponding to depths z_i and z_j($z_i > z_j$). In effect, this means that a large number of possible failure mechanisms, involving both local and overall failure, are checked. It is considered that this system provides a good approximation to limits which are both necessary and sufficient.

Limits for soils with cohesion and pore water pressure

The same arguments for redistribution may be followed through for both active and passive limits for soils with cohesion (c), pore water pressure (u) and effective wall pressure p' (where p' = p - u).

Between any two depths z_j and z_i ($0 \leq z_j \leq z_i$) the limits may be expressed as:

(active)

$$\int_{z_j}^{z_i} \left[u + K_a \left(u_j - u + \int_{z_j}^{z} \gamma dz + \left\{ K_{aj}\,p'_j - c_j K_{acj} \right\} \right) - cK_{ac} \right] dz \leq \int_{z_j}^{z_i} (u + p')dz \leq \qquad \text{(E6)}$$

(passive)

$$\int_{z_j}^{z_i} \left[u + K_p \left(u_j - u + \int_{z_j}^{z} \gamma dz + K_{pj}\,p'_j + c_j K_{pcj} \right) + cK_{pc} \right] dz \qquad \text{(E7)}$$

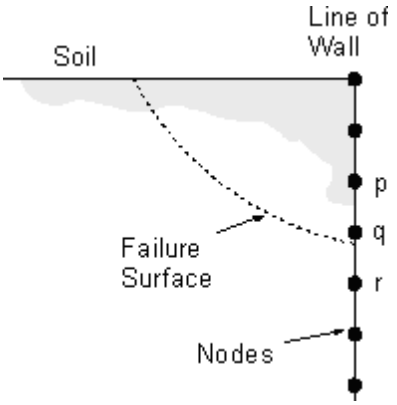
A further restriction is placed so that negative effective stresses are never used or implied. This is achieved by substituting zero for negative values of the expressions in brackets in the above inequalities.

In the passive pressure limit calculation it is generally not reasonable to impose the internal failure mechanism implied by Equation E7. The program therefore only enforces the limit implied by Equation E2, which states that the earth pressure integrated between the surface and depth z must not exceed the Rankine passive pressure integrated over the same depth. If Equation E7 indicates a failure however a small 'r' is included in the output table and the user must check that an internal failure mechanism would not occur.

© Oasys Ltd. 2014

8.4.3 Iterative Technique Adopted in Frew

In order to ensure that active and passive limits are not violated "displacement corrections" are computed for each node and added to the displacements derived from elastic analysis. They are displacements associated with plastic strain in the body of soil. When displacement corrections are used, the wall pressure at any node is still influenced, through the elasticity equations, by the movement of nodes below it but may be independent of its own movement and of the movement of nodes above it.



Suppose an active/passive failure occurs as shown above. The displacement corrections applied to ensure that the limits are not violated at node q will cause a change of stress but no displacement at node r, whilst at node p there will be a change of displacement but no change of stress. Effectively this means that movement is taking place at constant stress on the failure surface, whilst elastic conditions are still maintained, separately, in the blocks of material on either side of the failure surface.

The following procedure is used to achieve an iterative correction for wall pressures in the program.

The procedure starts at the top of the wall and works downwards.

1. For node i, calculate the correction (FORCOR) to the force between soil and wall required to return to the active/passive limit. If redistribution is specified, this correction will be a function of the pressures mobilised at nodes j above node i (i.e. j < i).

2. For node i, calculate approximately the displacement correction DCII (i) that would cause the force at i to change by FORCOR:

   DCII(i) = FORCOR/S (I,I)

   Where S(I,I) is the diagonal term of the soil stiffness matrix corresponding to node i.

3. For nodes j (j < i), calculate the displacement correction DCJI (i,j) that is required to prevent change of pressure at j when the displacement at i is corrected by DCII(i).

4. Repeat 1 to 3 for all nodes i.

5. For all nodes i calculate the total displacement correction

© Oasys Ltd. 2014

$$DCTOT(i) = DCII(i) + \sum_{j-1}^{i} DCJI(i,j)$$

6. Calculate the elastic soil force corrections from DCTOT x soil stiffness matrix, add these to the initial forces and recalculate the displacements ([DISP]) using the overall elastic system (the sum of the wall, strut and soil stiffness). The soil forces [F] acting at the nodes can then be recalculated as

   ([F] = [S] x ([DISP] - [DCTOT])

7. Repeat 1-6 iteratively to obtain convergence of DCTOT.

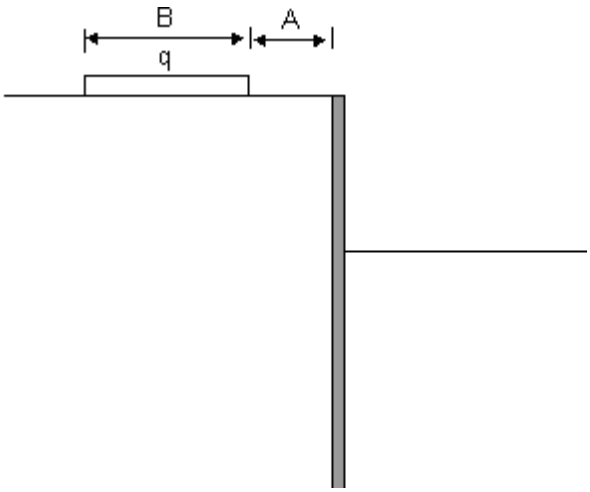
8.5 Active Pressures Due to Strip Load Surcharges

Considerable efforts have been made to formulate a relatively simple approximation to model the effect of a strip load on the active pressure limits.

Parametric studies were carried out using straight line and log spiral shaped failure surfaces and finite element work for soil that has constant properties with depth.

The ranges of variables considered were as follows;

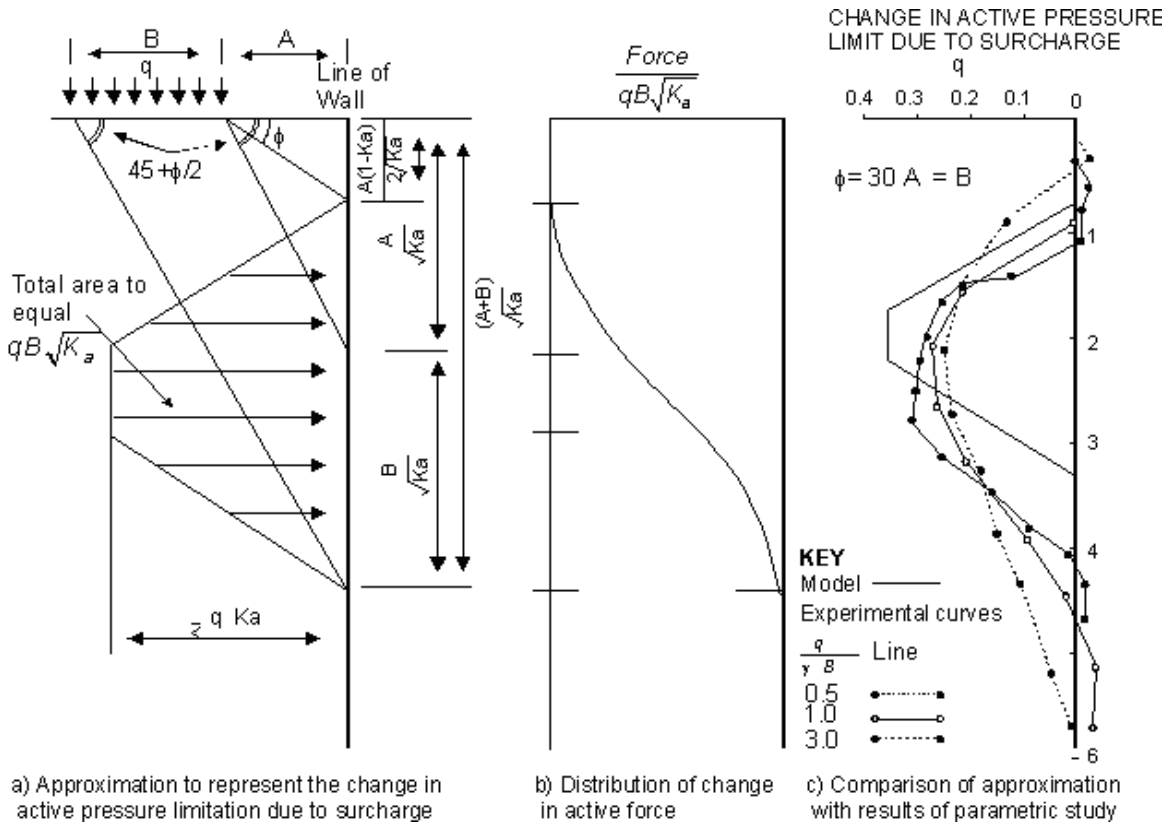
a) Ø' from 15 to 60,
b) q/B from 0.33 to 5 and
c) A/B from 0 to 2.



8.5.1 Application in Frew

It is important that any approximation that is chosen should be generally conservative. In the case of active pressure this can generally be achieved by over estimating the pressure near the top of the wall.

© Oasys Ltd. 2014



a) Approximation to represent the change in active pressure limitation due to surcharge

b) Distribution of change in active force

c) Comparison of approximation with results of parametric study

The results showed that the log spiral method, which is considered to be the best available approximation, usually gave very similar results to the straight line method.

From theoretical considerations the approximation illustrated above was developed to represent the increase in the active pressure limit, thus transferring the vertical pressure to a horizontal pressure on the wall.

This shows the shape of the pressure limit diagram and the criteria for calculation.

It should be noted that if the width of the load (B) is small, the diagram will become triangular. This pressure distribution is then used to modify the active pressure limit. Comparison of this distribution with the parametric studies suggests that it is generally conservative.

Variation of K_a with depth

If K_a varies with depth it is considered conservative to choose a mean value of K_a between any depth z and the level of the surcharge and then impose the criteria that the active force due to the surcharge, down to depth z be equal to the force derived from the above diagram. This is then subjected to the further limitation that the pressure never exceeds qK_{az} at any depth, where K_{az} is the active pressure coefficient at depth z.

© Oasys Ltd. 2014

8.5.2 Passive Pressures Due to Strip Load Surcharges

Frew calculates the increase in passive pressure due to a uniformly distributed load (udl), but can not make an allowance for strip load surcharges.

The program assumes that the passive pressure at depth z is equal to:

$$P_p = K_p \sigma_z' + 2c\sqrt{K_p} + u$$

where σ'_z is the vertical effective stress at depth z set equal to

$$\int_0^z \gamma dz + \sigma_{zudl} - u$$

σ_{zudl} is the sum of vertical pressure of all udl surcharges specified above z.

When there is wall friction $K_{pc} = \sqrt{K_p}$.

For strip load surcharges the user must adjust K_p (by adding additional soil layers if necessary) to allow for any increase in the passive pressure. This could be done using a series of trial failure surfaces to determine the passive pressure at any location.

Alternatively the user should check through and calculate the following requirements detailed here to derive the most suitable increase in the passive pressures.

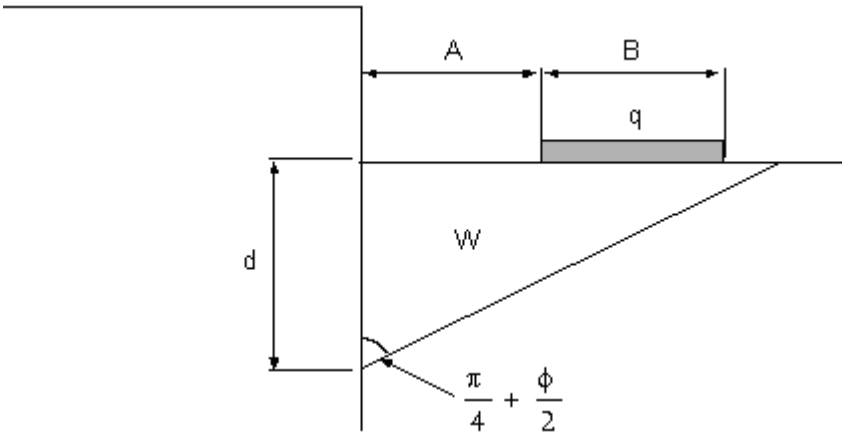
- Requirement 1 – General passive wedge.
- Requirement 2 – Check of the depth of the influence of the load.
- Requirement 3 – For a uniform surcharge.
- Requirement 4 – General application.

Note : The problem becomes more difficult if K_p varies with depth. A simple expedient would be to use equations specified in Requirement 4, with the appropriate value of K_p at each depth. This can be unsafe, however, if a soil with high K_p overlies a soil with low K_p. Requirement 1, which limits the total passive force effect to $qb\sqrt{K_p}$, could be violated for the less frictional soil.

© Oasys Ltd. 2014

8.5.2.1 Requirement 1

General passive wedge



Calculation

Consider a deep failure plane which will encompass the whole area of the strip load surcharge.

The following calculation assumes no wall friction. **Assume** that this is generally valid, even with wall friction.

Calculate the weight of the wedge (W);

$$W = \frac{1}{2}\gamma' d^2 \tan\left(\frac{\pi}{4} + \frac{\phi'}{2}\right) = \frac{1}{2}\gamma d^2 \sqrt{K_p}$$

Say the passive force due to the weight of the wedge (P_w) is

$$P_w = \frac{1}{2}\gamma' d^2 K_p = W\sqrt{K_p}$$

Thus, if W is increased by the effect of the surcharge, qB, the passive force will increase by $qB\sqrt{K_p}$.

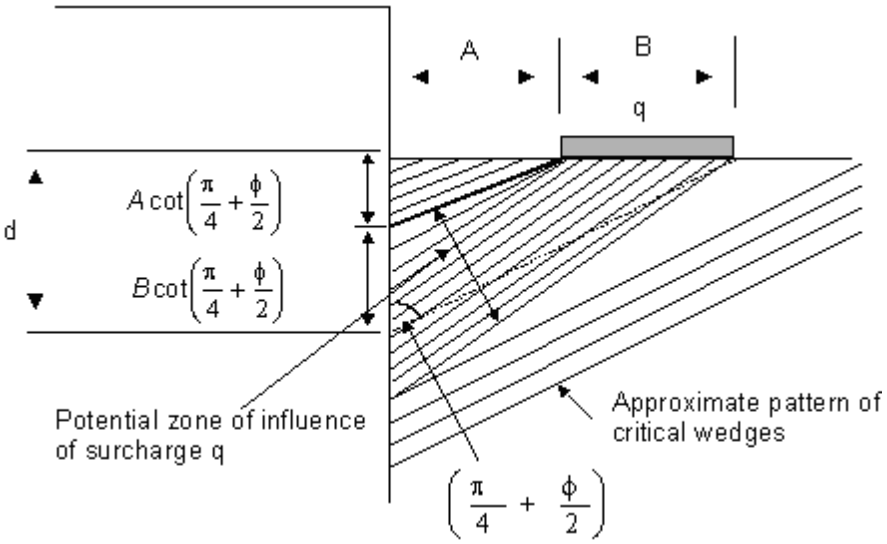
Therefore

$$P_p = (W + qB)\sqrt{K_p}$$

© Oasys Ltd. 2014

8.5.2.2 Requirement 2

Check of the depth of influence of the load.



Calculation

The effect of the surcharge (q) will not be felt above;

$$d = A\cot\left(\frac{\pi}{4}+\frac{\phi'}{2}\right)$$
$$= \frac{A}{\sqrt{K_p}}$$

This depth will be smaller in the presence of wall friction. In this case it is probably reasonable to use the same formula with a larger value of K_p.

Note : Extra passive force **may not** be fully applied by depth;

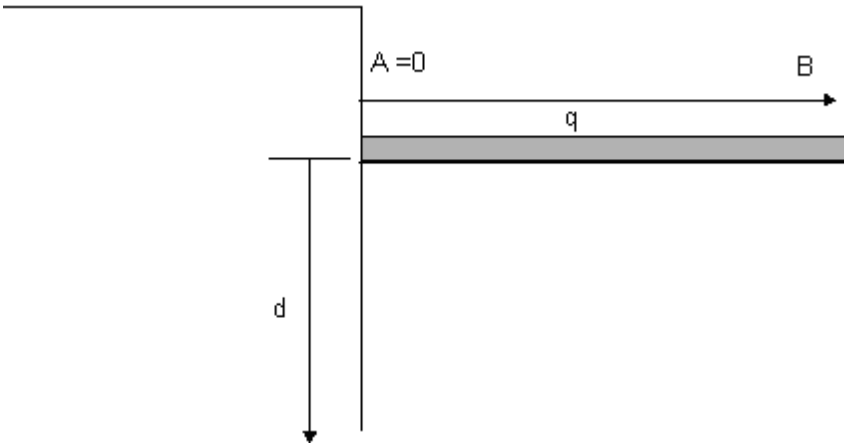
$$d = (A+B)\cot(45^0+\frac{\phi'}{2})$$
$$= \frac{(A+B)}{\sqrt{K_p}}$$

8.5.2.3 Requirement 3

For a uniform surcharge

A = 0 and B = ∞

© Oasys Ltd. 2014



The pressure due to the uniform load

$$P_{udl} = qK_p$$

It is unlikely that passive pressure increase for a strip load exceeds value for uniform surcharge, i.e. qK_p.

Note : The required total force P due to a uniform load

$$P_{udl} = qK_p\left[(A+B)\sqrt{K_p} - \frac{A}{\sqrt{K_p}}\right] = qB\sqrt{K_p}$$

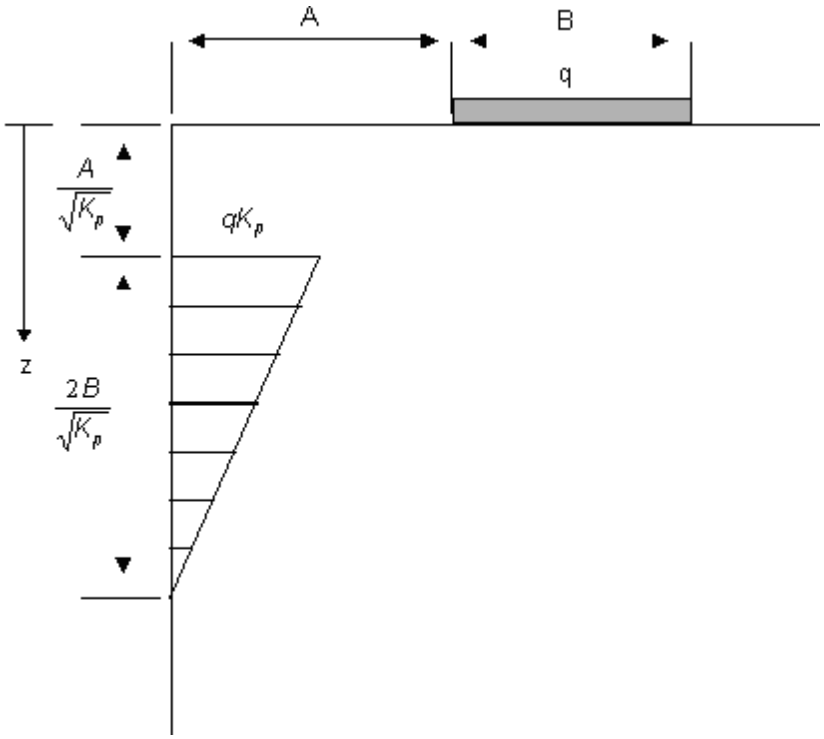
8.5.2.4 Requirement 4

General Application

To be safe, the effect of passive pressures should be placed rather low. Therefore the stress block shown below is considered to be generally suitable. The pressure can be expressed by the following equations:

Calculation

© Oasys Ltd. 2014



Depth criteria	Additional passive pressure due to load
$z < \frac{A}{\sqrt{K_p}}$	; $p_p = 0$
$\frac{A}{\sqrt{K_p}} < z < \frac{(A+2B)}{\sqrt{K_p}}$	; $p_p = qK_p \; 1 \; \frac{(z \; \frac{A}{\sqrt{K_p}})}{2B\sqrt{K_p}}$
$\frac{(A+2B)}{\sqrt{K_p}} < z$	; $p_p = 0$

8.6 Wall and Strut Stiffness Matrices

8.6.1 Wall Stiffness Matrices

The wall is modelled as a series of elastic beam elements, the stiffness matrix being derived using conventional methods from slope deflection equations. Considering a single beam element of length L and flexural rigidity EI spanning between nodes A and B, the moments (M) and forces (P) at nodes A and B can be expressed as functions of the deflections and rotation at the nodes i.e.:

© Oasys Ltd. 2014

$$\begin{bmatrix} M_A \\ M_B \end{bmatrix} = \frac{6EI}{L} \begin{bmatrix} 1 & -1 \\ 1 & -1 \end{bmatrix} \begin{bmatrix} \delta_A \\ \delta_B \end{bmatrix} + \frac{2EI}{L} \begin{bmatrix} 2 & 1 \\ 1 & 2 \end{bmatrix} \begin{bmatrix} \theta_A \\ \theta_B \end{bmatrix}$$

$$\begin{bmatrix} P_A \\ P_B \end{bmatrix} = \frac{12EI}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{bmatrix} \delta_A \\ \delta_B \end{bmatrix} + \frac{6EI}{L} \begin{bmatrix} 1 & 1 \\ -1 & -1 \end{bmatrix} \begin{bmatrix} \theta_A \\ \theta_B \end{bmatrix}$$

Where δ_A , δ_B and θ_A , θ_B, represent the deflections and rotations at nodes A and B respectively referred to the neutral axis of the beam. The above equations can be re-written in matrix form as:

$$[M] = [A]\,[\delta\,] + [B]\,[\theta] \qquad \text{(G1)}$$

and

$$[P] = [C]\,[\delta\,] + [A]^T\,[\theta] \qquad \text{(G2)}$$

where [A], [B] and [C] are functions of the element lengths and flexural rigidity (EI), and [δ] and [θ] are the nodal horizontal displacements and rotations.

If there are no moments applied to the wall [θ] can be eliminated to give

$$[P] = [S]\,[\delta\,] \qquad \text{(G3)}$$

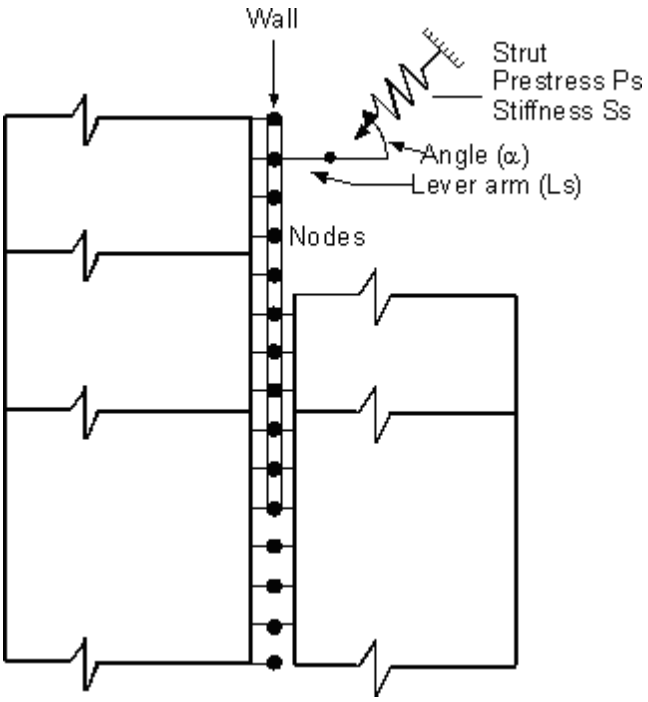
in which [S] is the wall stiffness matrix given by

$$[S] = [C] - [A]^T + [B]^{-1}\,[A] \qquad \text{(G4)}$$

© Oasys Ltd. 2014

8.6.2 Strut or Anchor Matrices

Struts or anchors can be installed at any node at any stage during the analysis.



As shown above the struts are specified as having a prestress force P_s and a stiffness S_s in terms of force/unit displacement.

A lever arm L_s and inclination α_s can also be specified to model the effect of a moment being applied to the wall by a strut or anchor. This feature can be used to model the effect of an inclined strut or anchor applying the force eccentrically to the wall section. If α_s is set 90Deg, it can also be used to model a moment restraint and an applied moment.

Based on the geometry defined above the force P and moment M applied at the node by the strut is given by

$$P = P_s \cos\alpha_s + \delta S_s \cos^2\alpha_s + \theta S_s L_s \cos\alpha_s \sin\alpha_s \quad \text{(G5)}$$

$$M = P_s L_s \sin\alpha_s + \delta S_s L_s \cos\alpha_s \sin\alpha_s + \theta S_s L^2_s \sin^2\alpha_s \quad \text{(G6)}$$

In these expressions d is the horizontal deflection of the node and q the rotation of the node since the introduction of the strut.

These equations can be written in the form of matrices that represented all struts currently acting on the wall as

$$[P] = [P_s \cos\alpha_s] + [S_{sh}]\,[\delta] + [S_{sc}]\,[\theta] \quad \text{(G7)}$$

$$[P] = [P_s L_s \sin\alpha_s] + [S_{sc}]\,[\delta] + [S_{sm}]\,[\theta] \quad \text{(G8)}$$

© Oasys Ltd. 2014

where the strut stiffness matrices are diagonal and equal to

$$[S_{sh}] = [S_s \cos^2\alpha_s] \quad \text{(G9)}$$

$$[S_{sc}] = [S_s L_s \cos\alpha_s \sin\alpha_s] \quad \text{(G10)}$$

$$[S_{sm}] = [S_s L_s \sin^2\alpha_s] \quad \text{(G11)}$$

The effect of the struts are incorporated into the analysis by matrix addition of the expressions given above to those given in equations G1 and G2, see General. Elimination of [θ] gives the following expression which is comparable to equation G3.

$$[P] = [D] + [S]\,[\delta] \quad \text{(G12)}$$

The new stiffness matrix for the wall $[S]$ including the effect of the struts, and the effect of the prestress $[D]$ are given by

$$[S] = [C] + [S_{sh}] - [[A] + [S_{sc}]]^T\,[[B] + [S_{sm}]]^{-1}\,[[A] + [S_{sc}]] \quad \text{(G13)}$$

$$[D] = [P_s \cos\alpha_s] + [[A] + [S_{sc}]]^T\,[[B] + [S_{sm}]]^{-1}\,[P_s L_s \sin\alpha_s] \quad \text{(G14)}$$

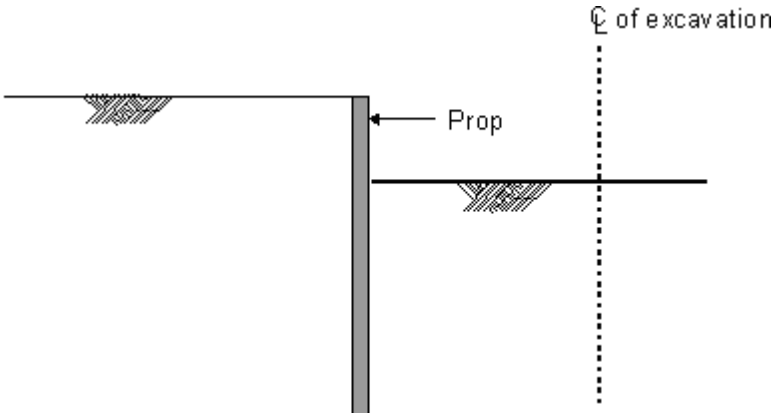
Of particular interest is the special case of a strut inclined at 90Deg to the wall for which equation (G6) reduces to

$$M = P_s L_s + \theta S_s L^2_s \quad \text{(G15)}$$

which allows moment restraint to be modelled at any node.

8.7 Modelling Axi-symmetric Problems Using Frew

Frew provides facilities for analysis of a plane strain excavation - i.e. an infinitely long trench.



Axi-symmetric problem

Many excavations are roughly square, and St John (1975) has shown that these can be modelled approximately as circular. For an axi-symmetric analysis the results apply to mid-side of the square.

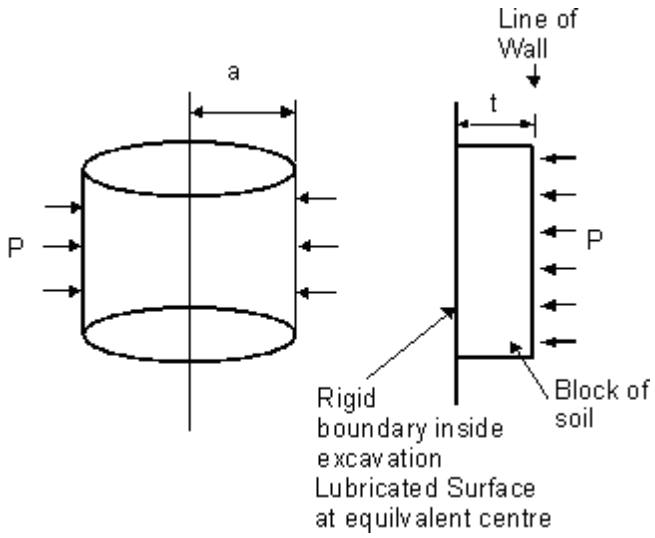
© Oasys Ltd. 2014

Once props have been inserted into an excavation, it makes little difference to the behaviour of the section being analysed whether the excavation is infinitely long or circular. This is because most of the strains which cause displacements are concentrated close to the props, and vertical arching within the soil governs the stress field. Furthermore, when the strength of the soil is fully mobilised in active and passive wedges, deformations are again localised and the geometry in plan is not too important.

However, situations can arise in which the plan geometry has a very significant effect on the magnitude of the movements. This is the case in heavily overconsolidated clays, for which the movements may be large before the strength is fully mobilised. As explained above, the effect is particularly important in computing movements **before the first props are inserted.**

8.7.1 Soil Inside the Excavation

Consider a cylinder of soil, radius a, inside a circular excavation. Consider the simplified case in which Young's Modulus, E, is constant with depth and the depth is large. To determine the horizontal stiffness, compare this with a block of thickness, t, and the same Young's Modulus, E.



Let the Poisson's ratios of the cylinder and the block be ν_c and ν_b respectively. For the same pressure p, the displacements d are:

$$\text{Cylinder}: \quad d = \frac{pa}{E}(1-\nu_c)$$

$$\text{Block}: \quad d = \frac{pt}{E}(1-\nu_b^2)$$

The displacements are equal when

$$t = a\frac{(1-\nu_c)}{(1-\nu_b^2)}$$

© Oasys Ltd. 2014

Note : In the "**Safe**" version of **Frew**, $\nu_b = 0.3$ and if the soil is undrained, take $\nu_c = 0.5$

Therefore for an undrained soil using the **Safe** model, the distance to the internal rigid boundary t;

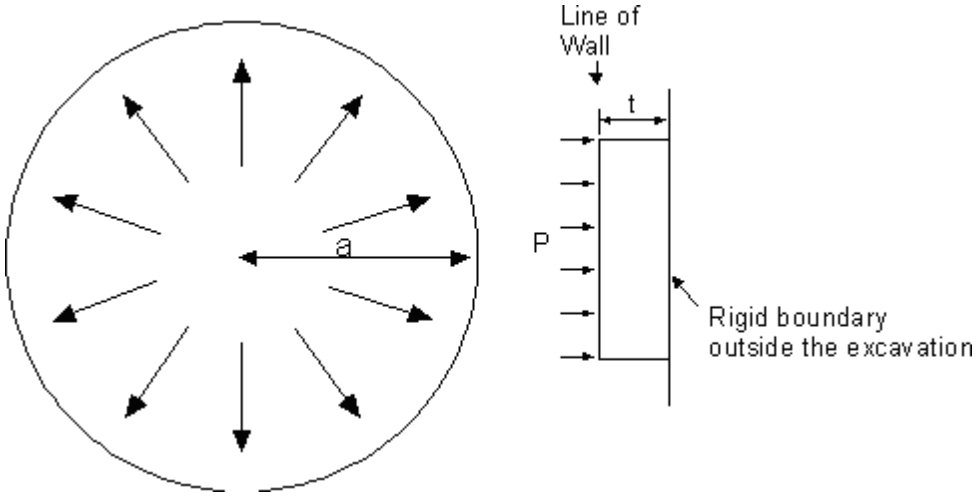
$t = (0.5 / 0.91)\, a = 0.55a$

a = radius of the actual excavation

8.7.2 Soil Outside the Excavation

The same simplifying assumptions can be made for the soil out side the excavation as for the soil inside.

Compare an expanding cylinder, radius a, with a block of thickness t



Poisson's ratios of the cylinder and the block are ν_c and ν_b respectively.

The displacements d are therefore:

$$\text{Cylinder} : \quad \mathrm{d} = \frac{pa}{E}(1+\nu_c)$$

$$\text{Block} : \quad \mathrm{d} = \frac{pt}{E}(1-\nu_b^2)$$

These are equal when

$$t = a\frac{(1+\nu_c)}{(1-\nu_b^2)}$$

Note : In the "**Safe**" version of **Frew**, $\nu_b = 0.3$ and if the soil is undrained, take $\nu_c = 0.5$

© Oasys Ltd. 2014

Therefore for an undrained soil using the **Safe** model, the distance to the external rigid boundary t ;

$t = (1.5 / 0.91)\, a = 1.65a$

a = radius of the actual excavation

8.7.3 Stiffness Varying with Depth

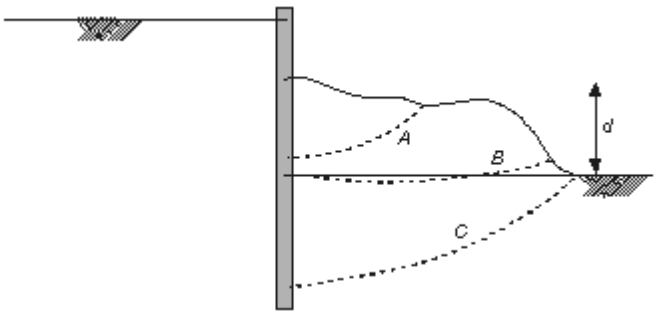
The analysis for axi-symmetric problems assumes that Young's modulus remains constant to great depth. In practice it usually increases with depth and the material becomes relatively rigid at a finite depth.

It is not obvious how this will affect the formulae given above, but a comparison of **Frew**, with a finite element run of **Safe** carried out for the British Library excavation, indicated that the formulae were reasonable approximations, giving displacements roughly 20% too large. However, this may be very dependent on the geometry of a particular problem; it is also dependent on the approximations used in **Frew** to represent elastic blocks of soil of finite length (see, Approximations used in the Safe Method).

8.8 Modelling Berms

Consider a berm of depth d, effective unit weight γ' and effective weight W.

The behaviour within the berm will be quasi-elastic until a failure plane develops. It will also be elastically connected to the ground beneath, until a failure develops. In the elastic phase, there will not be much difference in stress distributions between a berm and a uniform layer of the same height. In both cases, horizontal forces at the wall are transferred downwards by shear. The elastic behaviour, therefore can be modelled as if the berm were a complete layer of soil.



Berm Geometry

The figure shows three possible types of failure surface. These will develop at different stages, depending on the types of soil in the berm and in the ground beneath. (A particularly critical case occurs when a berm of frictional soil overlies frictionless ground.) It is possible, for example, that wall pressure above failure surface A will cause failure on surface B whilst surface A is still intact.

The user should propose equivalent values of K_p, K_{pc} and c - call them K^*_p, K_{pc} and c* - from which

© Oasys Ltd. 2014

passive pressures within the berm can be calculated.

$$P'_p = \gamma'_z K^*_p + 2c^* K_{pc}{}^*$$

The values of K^*_p and c* should be chosen such that the forces transferred through the berm will not be big enough to cause any failures, type A, B or even C.

The passive resistance of the ground beneath is also affected by the berm. At any depth a failure surface type C needs to be examined and a total passive force calculated. This passive force **includes** the horizontal forces transferred into the ground through the berm.

8.8.1 Rigorous Method

Therefore, berms may be treated as follows:

1. For elastic (and active) effects, treat as a full layer of soil.
2. For passive effects within the berm treat as full layer, but use K_p*, c*.
3. At the level of the excavation within the berm place a strip load surcharge with negative pressure (-q*) to remove the weight effect of the berm with properties:

   $$q^* = -\gamma' h$$

   A^* = average width of berm,
   B^* = distance to boundary, and
   $K_s{}^*$ = Kr of soil beneath berm.

Note that **Frew** disregards the effect of strip surcharges on limiting passive pressures. Therefore, this negative pressure does not change the limiting passive resistance of the ground below. It will, however, cause some movement which corresponds to the elastic effect of the excavation within the berm.

Also note that several surcharges could be used at various levels within the height of the berm. This could give a somewhat better approximation.

4. For the soil beneath the berm calculate amended K_p and c value as follows:
   - choose various levels below the berm, eg, points i, j etc.
   - at i determine critical failure surface C by using method of wedges (or using Oasys SLOPE) to give minimum passive force F_i. If straight-line wedges are used, the wall friction should normally be set to zero in this process.
   - calculate passive pressure p_i between base of berm (at level z_b) and point i (at level z_i) as

     $$p_i = (F_i - F_b) / (z_b - z_i)$$

     where F_b is the passive force within the height of the berm

© Oasys Ltd. 2014

$$F_b = \int_0^h (K_p^* \gamma' z + 2c^* \sqrt{K_p^*}) dz$$

which for a dry berm

$$= K_p^* \frac{\gamma h^2}{2} + 2c^* \sqrt{K_p^*} h$$

determine equivalent K_{pi} and c_i such that

$$K_{pi}\, \sigma'_{vi} + 2c_i K_{pi} = p_i - u_i$$

where u_j is the average pore pressure between points b and i.
σ'_{vi} is the average vertical effective stress (see Total and Effective Stress) between point b and i

$$\gamma h + \sigma_{zudl} + (\gamma / 2)(z_b - z_i) - u_i$$

where σ_{zudl} is the sum of all uniformly distributed surcharges above point i (usually none present in this case).

- at j determine critical failure surface D to give minimum passive force F_j :
- calculate passive pressure p_j between point i and point j (at level z_j):

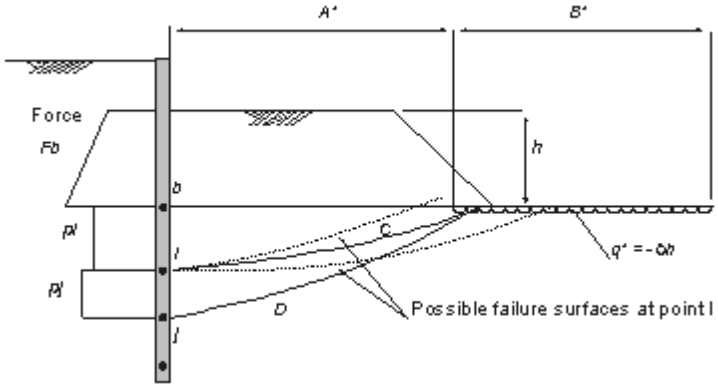
$$p_j = (F_j - F_i) / (z_i - z_j)$$

- determine equivalent K_{pj}, K_{pcj} and c_j such that

$$K_{pj}\sigma'_{vj} + 2c_j \sqrt{K_{pj}} = p_j - u_j$$

- continue down wall to base of problem.
- redefine soil zones and soil properties with K_{pi}, K_{pci}, c_i, etc, to base of problem.

© Oasys Ltd. 2014



Berm strip load surcharge and calculation of passive pressure beneath berm

The above procedures may get the active pressures slightly wrong, but that is of little consequence generally for berms. The procedures may cause the program to fail however with a message saying the active pressure is greater than the passive pressure at a node under the berm. If this occurs the active coefficient in the material under the berm should be reduced.

8.8.2 Simplified Procedure

This method is considered to be conservative, but simpler to use than that presented above. It relies on the fact that, in calculating passive (limiting) pressures, **Frew** only considers uniformly distributed surcharges (UDLs) and ignores the beneficial effects of strip surcharges.

1. For contact stresses between the berm and the wall, proceed as for the Rigorous Method above with Steps 1 and 2.
2. At the level of the base of the berm (the level of node b the figure above), apply a negative UDL surcharge $q_* = -\gamma h$
3. At the same level, apply a positive strip surcharge representing the berm itself. This will have a pressure equal to γh and width A*, as defined in the figure in Rigorous Method .

Below the berm, normal values of coefficients of active and passive pressure may now be used. Possible slips of Type C and D should be briefly reviewed, though it is unlikely that these will ever give a problem.

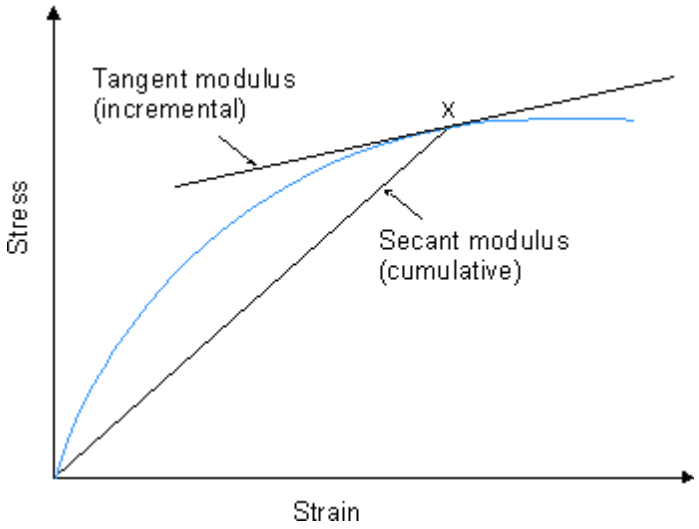
It is more likely that this method will be too conservative, especially in frictional materials ($\theta > 0$). This is because the benefit of the weight of the berm has been disregarded in calculating passive pressures below the level b. An allowance may be made for this by applying Steps 2 and 3 above at a slightly lower level, above which the weight of the full depth of the berm will be experienced by the ground. Better still, Steps 2 and 3 can be applied incrementally over a few depths, so that the adverse effect of the restricted length of the berm is applied gradually with depth.
When using this procedure it must be remembered that the force Fb is the passive force experienced within the height of the berm, which is transferred as an adverse horizontal shear force to the ground beneath. Therefore the additional passive resistance (force) in the ground beneath the berm, due to the presence of the berm, may be taken as $W\sqrt{K_p} - F_b$, where W is the weight of the berm. This formula will need refinement if K_p varies with depth.

© Oasys Ltd. 2014

8.9 Creep and Relaxation

There is often a requirement to model **creep** in **Frew**. Usually this refers to the creep of concrete structures which structural engineers normally represent by a change of Young's modulus. However, it cannot be represented so simply in **Frew**.

Elastic moduli are defined as ratios of stress to strain. The stresses and strains used in this ratio may be either cumulative values (starting from zero stress and zero strain), or incremental values.



The above shows these two possibilities for a point X on a non-linear stress-strain curve.

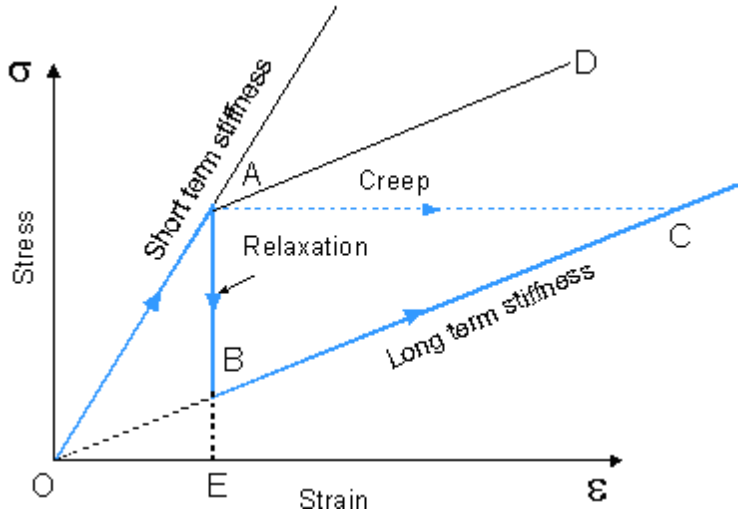
The modulus in terms of cumulative stress and strain is commonly referred to as a secant modulus, whilst the modulus related to a small increment of stress and strain at point X is a tangent modulus.

Note : It is also possible to have an incremental secant modulus which approximates to the tangent modulus for small increments.
In describing the change of stiffness from short-term to long-term as a change of stiffness modulus, structural engineers are referring to secant moduli. But it is important to realise **that Frew uses tangent (incremental) moduli as its basic data**. This means that merely changing stiffness moduli, when nothing else is changed, will have no effect at all.

8.9.1 Changing From Short Term to Long Term Stiffness

The following shows the type of stress-strain curve required for a change from short-term to long-term stiffness.

© Oasys Ltd. 2014



Creep and relaxation

The high short-term stiffness on OA is required to drop to the lower long-term stiffness on line OBC. Consider an element of structure which in the short-term has been stressed to Point A. In the course of time, its state will move to be somewhere on line BC. If it is in a situation in which there is no change of strain during this change, stresses will simply relax and it will move to point B. If, on the other hand, the load on the element can not change, it will creep and move to point C. Thus relaxation and creep are different manifestations of the same phenomenon. It is easier to think about the working of **Frew** in terms of relaxation than of creep.

In **Frew**, if an element is at point A and the only change made is to change the Young's modulus in the data, further behaviour will proceed along line AD. This does not represent creep or relaxation. Somehow the program must be informed that even if nothing moves, stresses will change from point A to point B. If these new stresses are no longer in equilibrium, the program will then respond by further strains and the stress state will move up line BC.

Creep effects on bending stiffness (EI) in **Frew** can be modelled directly by using wall relaxation.

Relaxation calculation

The relaxation percentage is defined as (AB/AE)*100. For example for a concrete, the short-term and long-term stiffness' are taken as 30 and 20GPa respectively. The relaxation percentage would be

[(30 - 20) / 30] * 100 = 33.3%

At the same time the value of wall stiffness will be reduced to 20GPa.

Supporting struts and slabs

Creep in supporting struts or slabs can also be modelled. Use two struts at the required level which give the correct short-term stiffness when combined. Then remove one of them to obtain the long-term effect.

Note : In **Frew**, when a strut is removed, the force associated with it is also removed.

© Oasys Ltd. 2014

8.10 Undrained to drained behaviour - Manual Process

To effect a switch from undrained to drained behaviour without using **Frew** to calculate the undrained pore pressures, the following procedure should be carried out.

The user must first calculate and specify the profile of undrained pore pressures, then change them to the drained values.

Note: It is not sufficient merely to change to effective stress parameters and impose the final (drained) distribution of pore pressure.

The following procedure has been found to be satisfactory.

Note : In the **Frew** tabular output data

P_e = apparent horizontal (effective) stress

V_e = apparent vertical (effective) stress

u = pore water pressure, specified by the user.

1. Tabulate the values of vertical and horizontal effective stresses given by **Frew** for Stage 0 or the previous drained stage, for both the left and right sides of the wall. Add the user input values of pore pressure u to obtain the total stresses.
2. Tabulate the corresponding values for the final undrained stage in the analysis.
3. Calculate the change in total horizontal $\Delta\sigma_h$ and vertical $\Delta\sigma_v$ stresses between the two stages.
4. Calculate the "actual" change in pore water pressure due to movement in the wall between the two stages.

   This demands an understanding of the undrained stress path. For soils which are at yield in shear, Skempton's pore pressure parameter A would be useful if its value can be assessed. If the soils have not reached yield, a modified value of A is required.

   $\Delta u = B\Delta\sigma_h + A(\Delta\sigma_v - \Delta\sigma_h)$ Skempton's equation.

   In stiff clays, it may be assumed that the mean normal effective stress remains constant during shearing. This assumption would not necessarily be appropriate to other soils and should always be reviewed carefully. In the plane strain conditions of **Frew**, it amounts to:

   $\Delta u = (\Delta\sigma_h + \Delta\sigma_v) / 2$

5. Add Δu values to the user input values of *u* at stage 0.
6. Input this adjusted u value into the **Frew** analysis.

   Three additional stages (A, B and C) are now required to complete the transition between undrained and drained behaviour.

7. Stage A. The new **pore pressure profile** should now be introduced into the next stage, without changing the (undrained) strength criteria of the soils and **keeping Kr = 1.** This causes **Frew** to recalculate effective stresses, but total stresses are unchanged and no movement occurs.
8. Stage B. The **soil strength parameters** should then be changed to drained values,

© Oasys Ltd. 2014

retaining the undrained pore pressures.

This provides a check as, if the above procedure has been carried out correctly, this step should have no effect and could be omitted. However, movement will occur if the specified pore pressures and effective strength parameters are not consistent with the computed horizontal effective stresses. Since the horizontal stresses are consistent with the undrained strengths, this would imply inconsistency between the specified pore pressures, drained and undrained strengths.

In more complex analyses it is possible that assumptions made in the above process may cause small amounts of movement to occur in Stage B. If this occurs the user should review the modelling of the problem and the strength criteria to ensure that results are reasonable.

9. Stage C. Finally, the **pore pressure profile** should be changed to its long-term (drained) values with **Kr < 1**, as for drained behaviour. This will cause changes in horizontal total stresses and movement will be needed to restore equilibrium.

8.10.1 Undrained to Drained Examples

The following provides an example of a manually applied transition between undrained and drained materials.

Soil Properties

Soil	E0 (kPa)	Unit wt. (kN/m^3)	Ko	Ka	Kp	Kac	Kpc	Kr	c0	Y0	Gradient (kPa/m) cE	
Undrd	50000	18	1.0	1.00	1.0	2.00	2.00	1.00	100	50.0	10	2000
Drained	35000	18	1.0	0.15	7.0	0	0	0.25	0	50.0	0	2000

Stage Data

Stage 0	Initial drained conditions, with PWP specified.
Stage 1	Undrained materials and excavation to 48.5m OD
Stage 2	Excavation to 42.5m OD, insert strut 1, piezometric pressure profile specified on passive side to create submerged excavation.
Stage 3 (Stage A)	New pore water pressure profile, calculated by user for undrained conditions.
Stage 4 (Stage B)	Changed to drained parameters.
Stage 5 (Stage C)	Pore water pressure to drained profile.

Calculation procedure

In this example it is assumed that

$$\Delta u = (\Delta\sigma_v + \Delta\sigma_h) / 2$$

© Oasys Ltd. 2014

Note: Users should be aware that as K_o is used to calculate P_e in stage 0, total horizontal stress in this stage should be calculated by adding K_o*u to P_e. In any other undrained stage K_a / K_p are used and are equal to 1, therefore total horizontal stress can be calculated by simply adding u to P_e.

Pore water Pressure Calculations

Active Pressures

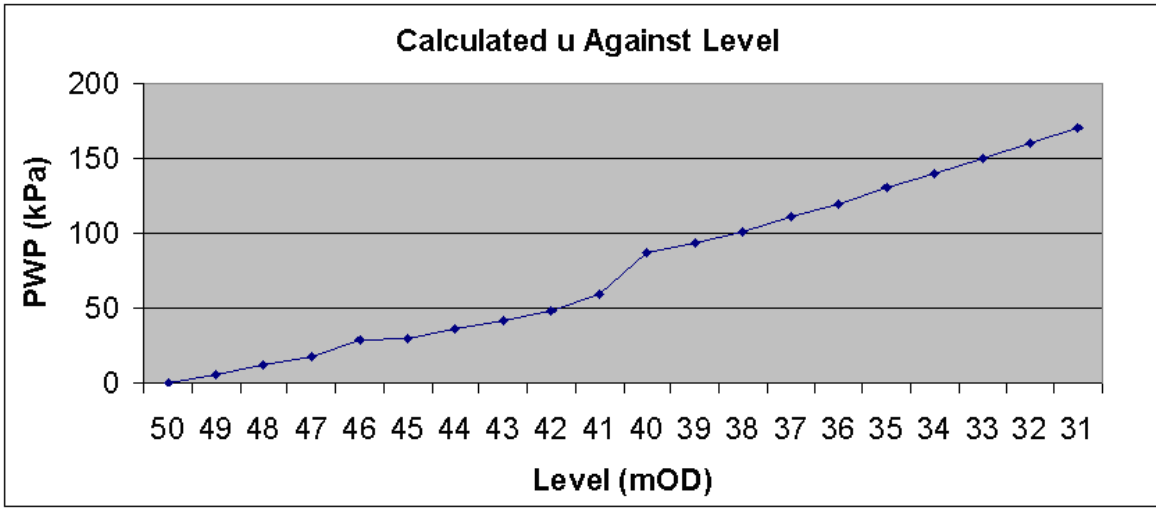
STAGE 0 PARAMETERS

NODE NO	LEVEL	Effective Pe	Effective Ve	User Spec u	Total Pe	Total Ve
1	50	0	0	0	0	0
2	49	8	8	10	18	18
3	48	16	16	20	36	36
4	47	24	24	30	54	54
5	46	23	32	40	63	72
6	45	40	40	50	90	90
7	44	48	48	60	108	108
8	43	56	56	70	126	126
9	42	64	64	80	144	144
10	41	72	72	90	162	162
11	40	80	80	100	180	180
12	39	88	88	110	198	198
13	38	96	96	120	216	216
14	37	104	104	130	234	234
15	36	112	112	140	252	252
16	35	120	120	150	270	270
17	34	128	128	160	288	288
18	33	136	136	170	306	306
19	32	144	144	180	324	324
20	31	152	152	190	342	342

© Oasys Ltd. 2014

LAST UNDRAINED STAGE PARAMETERS - Stage 2

NODE NO	LEVEL	Effective Pe	Effective Ve	User Spec u	Total Pe	Total Ve	Total Del Pe	Total Del Ve	DEL u	u
1	50	0	0	0	0	0	0	0	0	0
2	49	0	8	10	10	18	-8	0	-4	6
3	48	0	16	20	20	36	-16	0	-8	12
4	47	0	24	30	30	54	-24	0	-12	18
5	46	0	32	40	40	72	-23	0	-11.5	28.5
6	45	0	40	50	50	90	-40	0	-20	30
7	44	0	48	60	60	108	-48	0	-24	36
8	43	0	56	70	70	126	-56	0	-28	42
9	42	0	64	80	80	144	-64	0	-32	48
10	41	10.7	72	90	100.7	162	-61.3	0	-30.65	59.35
11	40	54.6	80	100	154.6	180	-25.4	0	-12.7	87.3
12	39	54.8	88	110	164.8	198	-33.2	0	-16.6	93.4
13	38	58.6	96	120	178.6	216	-37.4	0	-18.7	101.3
14	37	65.4	104	130	195.4	234	-38.6	0	-19.3	110.7
15	36	71.4	112	140	211.4	252	-40.6	0	-20.3	119.7
16	35	80.6	120	150	230.6	270	-39.4	0	-19.7	130.3
17	34	88.5	128	160	248.5	288	-39.5	0	-19.75	140.25
18	33	96.4	136	170	266.4	306	-39.6	0	-19.8	150.2
19	32	104	144	180	284	324	-40	0	-20	160
20	31	112	152	190	302	342	-40	0	-20	170



Passive Pressures

© Oasys Ltd. 2014

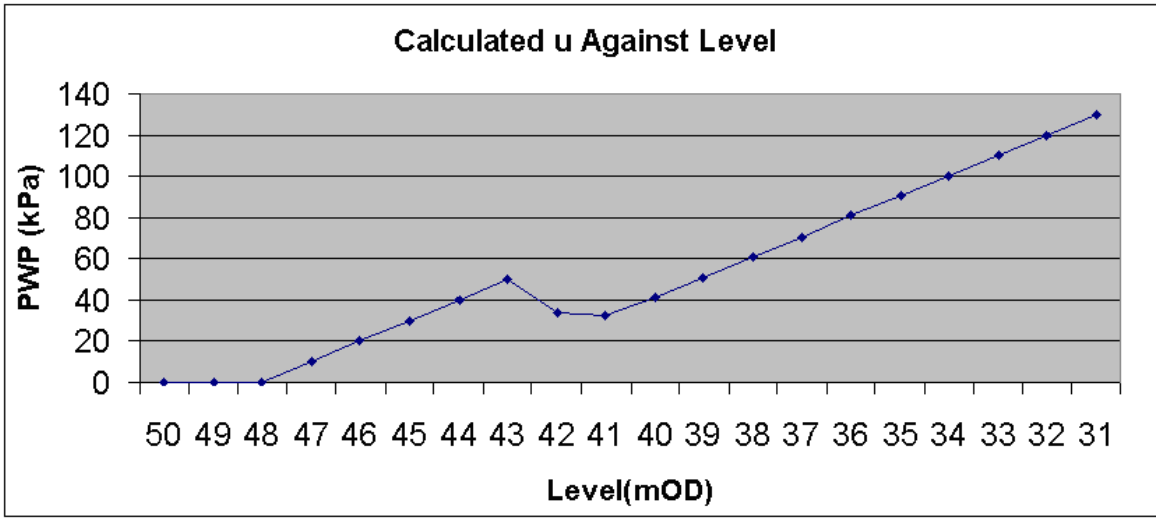
STAGE 0 PARAMETERS

NODE NO	LEVEL	Effective Pe	Effective Ve	User Spec u	Total Pe	Total Ve
1	50					
2	49					
3	48					
4	47					
5	46					
6	45					
7	44					
8	43					
9	42	64	64	80	144	144
10	41	72	72	90	162	162
11	40	80	80	100	180	180
12	39	88	88	110	198	198
13	38	96	96	120	216	216
14	37	104	104	130	234	234
15	36	112	112	140	252	252
16	35	120	120	150	270	270
17	34	128	128	160	288	288
18	33	136	136	170	306	306
19	32	144	144	180	324	324
20	31	152	152	190	342	342

LAST UNDRAINED STAGE PARAMETERS

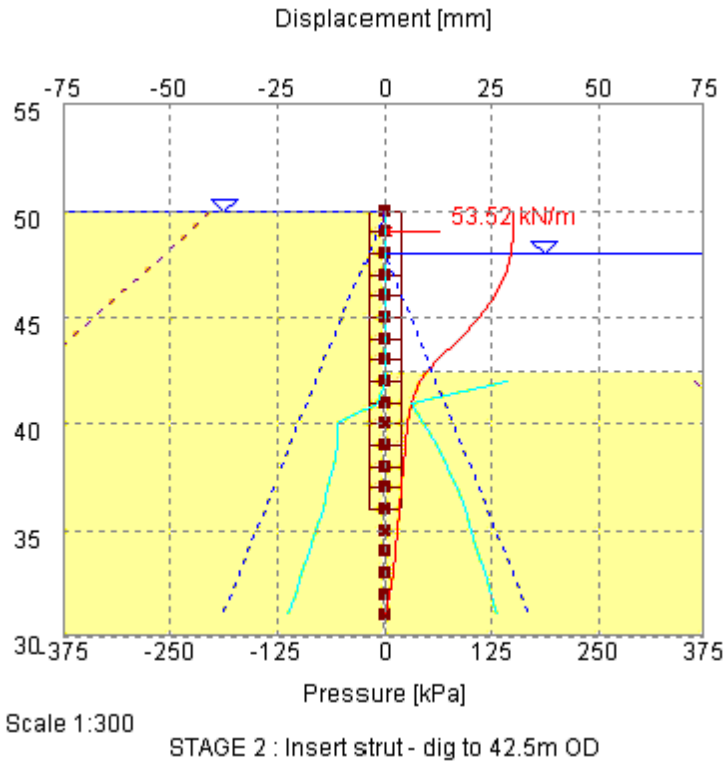
NODE NO	LEVEL	Total Pe	Total Ve	User Spec u	Total Del Pe	Total Del Ve	DEL u	u
1	50						0	0
2	49						0	0
3	48						0	0
4	47						10	10
5	46						20	20
6	45						30	30
7	44						40	40
8	43						50	50
9	42	131.6	64	0	-12.4	-80	-46.2	33.8
10	41	127.4	82	0	-34.6	-80	-57.3	32.7
11	40	142.5	100	0	-37.5	-80	-58.75	41.25
12	39	159.7	118	0	-38.3	-80	-59.15	50.85
13	38	177.4	136	0	-38.6	-80	-59.3	60.7
14	37	194.8	154	0	-39.2	-80	-59.6	70.4
15	36	213.7	172	0	-38.3	-80	-59.15	80.85
16	35	230.8	190	0	-39.2	-80	-59.6	90.4
17	34	248.6	208	0	-39.4	-80	-59.7	100.3
18	33	266.3	226	0	-39.7	-80	-59.85	110.15
19	32	283.9	244	0	-40.1	-80	-60.05	119.95
20	31	302	262	0	-40	-80	-60	130

© Oasys Ltd. 2014



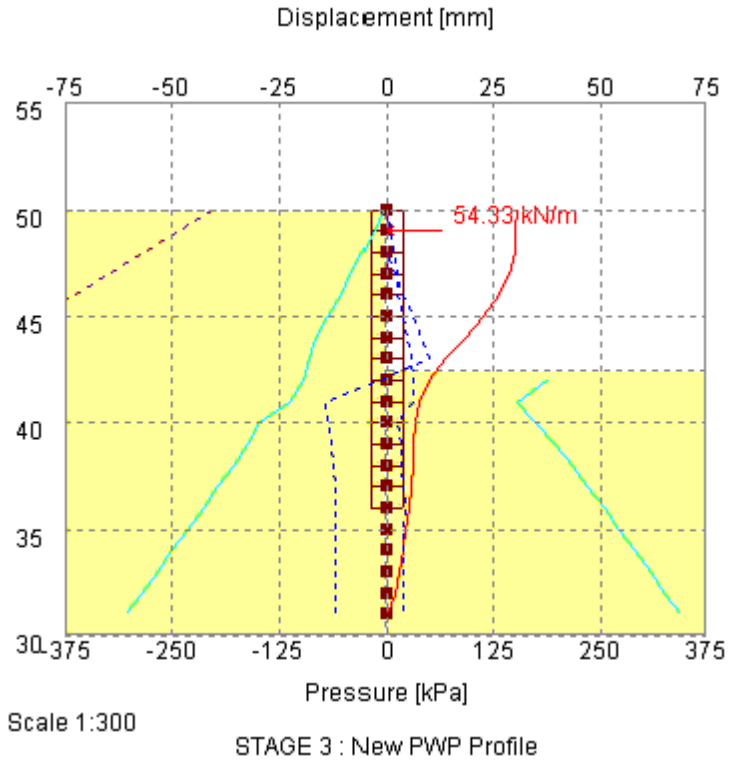
Graphical Results

Last Undrained Stage

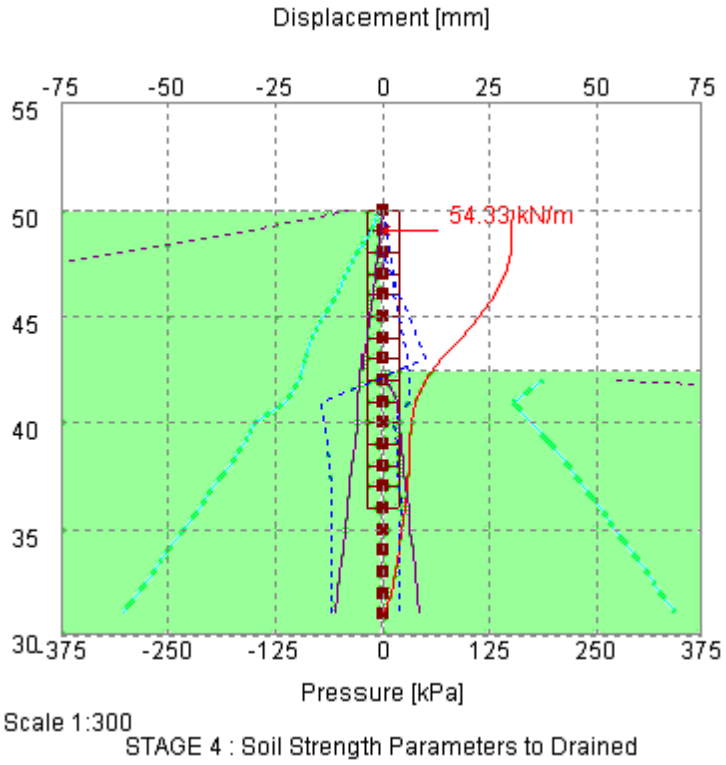


Stage A – Adjusted PWP

© Oasys Ltd. 2014

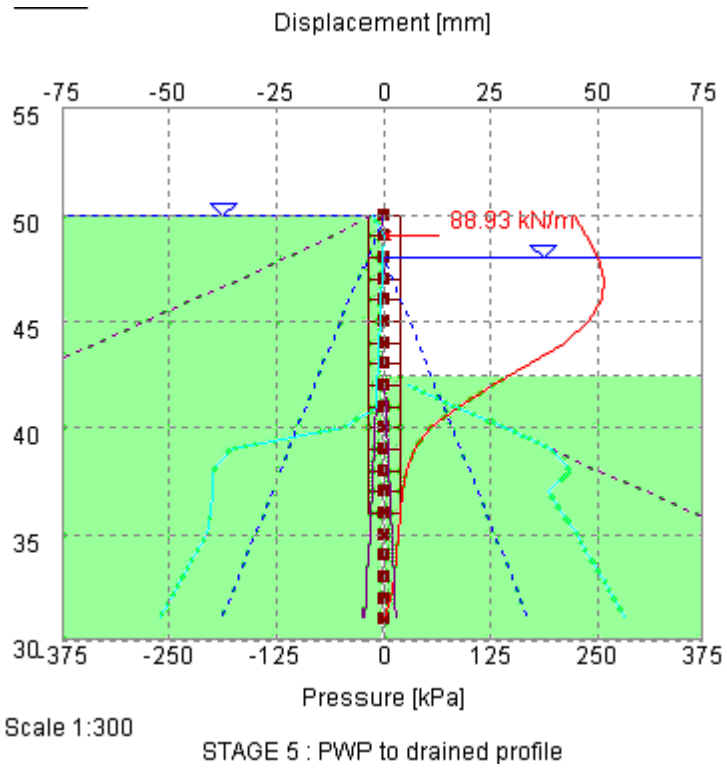


Stage B – Switch to Drained parameters



Stage C – Revert to Drained PWP Profile

© Oasys Ltd. 2014



9 List of References

9.1 References

British Standards Institute (2004) (+A1 2013). Design of Structures for Earthquake Resistance - Part 1: General rules, seismic actions and rules for buildings. BS EN 1998-1:2004 + A1:2013.

British Standards Institute (2004). Design of Structures for Earthquake Resistance - Part 5: Foundations, retaining structures and geotechnical aspects. BS EN 1998-5:2004.

British Standards Institute (2011). Recommendations for the design of structures subject to traffic loading to BS EN 1997-1:2004. Published Document PD6694-1:2011.

Broms B B (1972). Stability of flexible structures. General Report, 5th Euro Conf SMFE, Vol 2, Madrid.

Mononobe N and Matsuo M (1929). On the determination of earth pressures during earthquakes. Proceedings of the World Engineering Congress, Vol. 9, 179-187, 1929.

Okabe S (1926). General theory of earth pressure. Journal of the Japanese Society of Civil Engineers, Tokyo, Japan, Vol. 12 , No. 1.

Pappin J W, Simpson B, Felton P J, and Raison C (1985). Numerical analysis of flexible retaining walls. Proc NUMETA '85, University College, Swansea, pp 789-802.

© Oasys Ltd. 2014

Pappin J W, Simpson B, Felton P J, and Raison C (1986). Numerical analysis of flexible retaining walls. Symposium on computer applications in geotechnical engineering. The Midland Geotechnical Society, April.

Poulos H G (1971). Behaviour of laterally loaded piles : I Single piles. Proc ASCE JSMFE, 97, 5, 711-731.

Seed H and Idriss I N (1970). Soil Moduli and Damping Factors for Dynamic Response Analyses. *Report no. EERC 70-10*. University of
California: California.

Simpson B. (1994) Discussion, Session 4b, 10th ECSMFE, Florence, 1991, Vol 4, pp1365-1366.

St John H D (1975). Field and theoretical studies of the behaviour of ground around deep excavations in London Clay. PhD Thesis, University of Cambridge.

Vaziri H, Simpson B, Pappin J W, and Simpson L (1982). Integrated forms of Mindlin's equations. Géotechnique, 22, 3, 275-278.

Wood J H (1973). Earthquake-Induced Soil Pressures on Structures. PhD Thesis, California Institute of Technology.

Wood J H & Elms D G (1990). Seismic Design of Bridge Abutments and Retaining Walls. Road Research Unit Bulletin 84 Volume 2, RRU, Transit New Zealand, Wellington.

10 Brief Technical Description

10.1 Suggested Description for Use in Memos/Letters, etc

Frew is a program used to analyse the behaviour of flexible retaining walls. It predicts the displacement, shear forces, and bending moments of the wall and the earth pressures each side of the wall resulting from a series of actions. These actions include excavation, filling, dewatering, changing soil or wall properties and applying or removing struts, anchors or surcharges. The program models the soil as an elastic continuum and allows for soil failure by restricting the earth pressures to lie within the active or passive limits and also includes the effect of arching.

10.2 Brief Description for Inclusion in Reports

The following pages contain a summary of the analysis method used by **Frew**. It is intended that they can be copied and included with calculations or reports as the need arises.

Frew is a program to analyse the soil structure interaction problem of a flexible retaining wall, for example a sheet pile or diaphragm wall.

The wall is represented as a line of nodal points and three stiffness matrices relating nodal forces to displacements are developed. One represents the wall in bending and the others represent the soil on each side of the wall. The soil behaviour is modelled using one of three methods:

1. **"Safe" flexibility method** - the soil is represented as an elastic solid with the soil stiffness matrices being developed from pre-stored stiffness matrices calculated using the

© Oasys Ltd. 2014

"**Safe**" finite element program. This method is ideally limited to a soil with linearly increasing stiffness with depth, but empirical modifications are used for other cases.

2. **Mindlin method** - the soil is represented as an elastic solid with the soil stiffness based on the integrated form of the Mindlin Equations. This method can model a wall of limited length in plan but is ideally limited to a soil with constant stiffness with depth but again empirical modifications are used for other cases.
3. **Subgrade reaction method** - the soil is represented as a series of non interactive springs. This method is considered to be unrealistic in most circumstances.

The program analyses the behaviour for each stage of the construction sequence. At each stage it calculates the force imbalance at each node imposed by that stage and calculates displacement and soil stresses using the stiffness matrices. If the soil stresses are outside the active or passive limiting pressures correction forces are applied and the problem solved iteratively until the stresses are acceptable. Allowance can be made for arching within the soil body when calculating the active and passive limiting pressures.

The following input parameters are included in the analysis:

- problem geometry including dig depths, distances to remote boundaries
- wall profile bending stiffness and creep
- soil stratification, strength, density and stiffness
- struts (or anchors) including prestress, stiffness, inclination and a lever arm (to represent rotational fixity).
- surcharges including depth and extent
- groundwater levels and pore pressures each side of wall.

The program gives results for earth pressures, shear forces and bending moments in the wall, strut forces and displacements. These are presented in tabular form and can be plotted diagrammatically. In addition the number of iterations, the displacement error between successive interactions and the maximum earth pressure error are output.

Full details of the assumptions and analysis methods are included in the following paper.

Pappin J W, Simpson B, Felton P J, and Raison C (1986). "Numerical analysis of flexible retaining walls". Symposium on computer applications in geotechnical engineering. The Midland Geotechnical Society, April.

11 Manual Example

11.1 General

The data input and results for the manual example are available to view in **Frew** data file (.fwd) format or pdf format in the 'Samples' sub-folder of the program installation folder. The example has been created to show the data input for all aspects of the program and does not seek to provide any indication of engineering advice.

Screen captures from this example have also been used throughout this document.

This example can be used by new users to practise data entry and get used to the details of the program.

© Oasys Ltd. 2014

Index

A

Active and Passive Limits 8
Active earth pressure
  Axi-symmetric problems 142
  Limits 11, 12, 126, 128, 132
  Output 109
  Pressures 15, 34, 127
  Surcharges 45, 133
  Surcharges: 133
  Tolerance 73
Analysis
  Methods of 1
  Procedure 11
Anchors 43, 141
Angle of friction 83
Assembling data 24
Axis
  Graphical output 109
Axi-symmetric Problems 142, 145

B

Backfill
  Effects of 16, 24
  Modelling of 59
Batch plotting 112
Bending moments 11
berm 145
berms 145
Bitmaps
  In titles window 33
Boundary 67
  Distances 67
  Horizontal rigid 37, 67
  Vertical 67, 118
boundary distances 67, 74

C

Components of the User Interface 3
Convergence Control 72
Copy (graphics) 109
Creep 149

D

Damping coefficient 73
Data Checking 101
Data Entry 74
Deflection
  Calculation 139
  Graphical representation 109
Direct Kp factors 47
Displacement
  Maximum incremental 74
  Tolerance of 73
Drained materials 16, 18, 19, 21, 24, 34, 44, 52, 152

E

Earth pressure at rest 18, 34
Effective stress 18
element edge loads 82
Example
  Analysis procedure 11, 12, 13
  Manual 24
Excavation
  Effects of 16, 37, 45, 123
  Modelling of 59, 142
Export 74

F

Factors of safety 47
File
  New FREW file 24
first stage material 85
Fixed and Free solution 68, 122
Fixed Earth Mechanisms 4
Fixed or Free solution 68
Free Earth Mechanisms 5
Frew Toolbar 3

G

'g' 85

© Oasys Ltd. 2014

Gateway 3
Geometry 81
Global Data 24, 32, 33, 34, 37, 41, 52
global softening 71
Graphical Output 3, 109
Graphics Toolbar 3
Groundwater 74, 87
  Hydrostatic 62
  Piezometric 62

I

Inserting Stages 53
Iterations
  Number of 73

K

K0 85

L

Lever arm 82
linear elasttic 83

M

Material Properties 34, 52
Materials 83
Mesh 81
Mindlin method
  Accuracy of 124
  Application in FREW 123
  Basic model 13, 67, 123
Mohr-Coulumb 83

N

New Stages 51, 52
Nodes 1, 11, 24, 37, 56, 60, 107, 113

P

Partial Factors 47
Passive earth pressure 34, 135
  Limits 11, 126, 128, 132
  Output 34, 109
  Pressures 15, 34
  Surcharges 45
Passive earth pressure:Surcharges 135
Passive softening 71
passive softening depth 71
Poisson's ratio 83
pore pressure 87
Pressure 73
PressureTolerance of 73

R

Radius of influence 87
Redistribution 15, 69, 126
Relaxation 24, 60, 68, 149
Restraints 81
Results 33, 101, 107, 108, 109, 160
Rigid boundary 67
Runs 79

S

SAFE method 1, 12, 68, 113
  Accuracy of 13
  Approximations 113
  Fixed or free 68
Save Metafile 109
Scale
  Engineering 109
Shear Force 1, 107, 109
Soil strength factors 47
Soil Zones 56
Stage
  Changing titles 53, 54
  Construction 1
  Data 51
  Deleting 53
  Editing 52
  Stage 0 11, 19, 24, 52
Stages 79
Standard Toolbar 3
Strip Loads 44, 45, 135
  Active pressures 133
  Passive pressures 135
Struts 82
  Levels 37

© Oasys Ltd. 2014

Struts 82
Properties 1, 41, 139, 141
Sub-grade Reaction 14, 67, 118
Surcharge factors 47
Surcharges 44, 82
Strip loads 15, 44, 135
Strip loads: 133
Uniformally distributed loads 18, 44, 45

T

Table View 3
Tabular Output 3
Tabulated Output 104
Titles 32, 54
Toolbar 3

U

Uniformally distributed loads 44, 45
Units 33
Unsupported features 88
User defined factors 47
User Interface 3

V

validation 74
void 83

W

Wall 1, 83
Data 11, 60
Deflection 109
Friction 13, 15, 122
Geometry 1, 24, 60
Relaxation 24, 60, 68
Stiffness 1, 37, 41, 43, 44, 45, 51, 52, 53, 54, 55, 56, 60, 139
Stiffness: 52
wall data 74
Wall Friction 122

Y

Youngs modulus 68

Z

Zoom Facility 109

© Oasys Ltd. 2014

Endnotes 2... (after index)

© Oasys Ltd. 2014